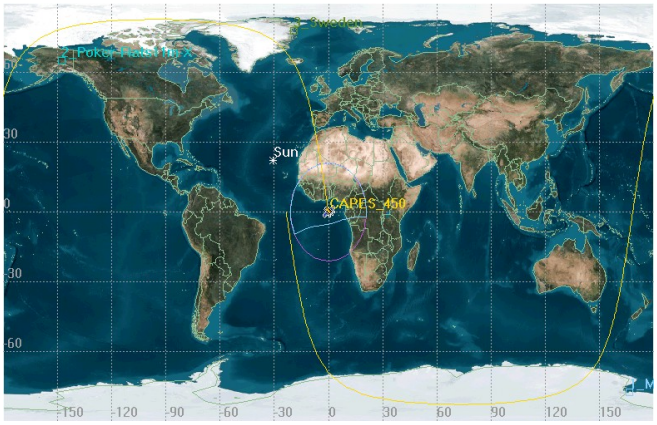
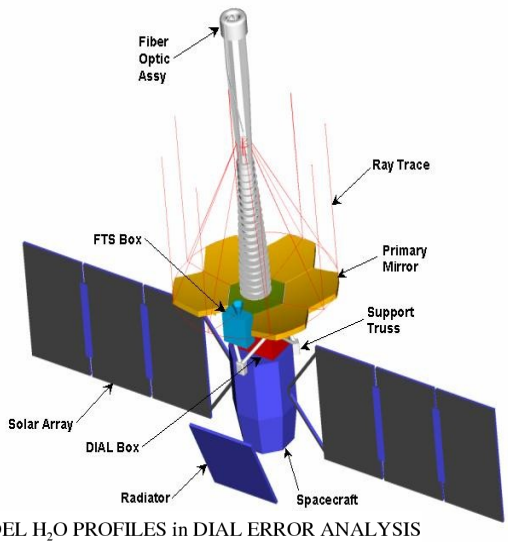
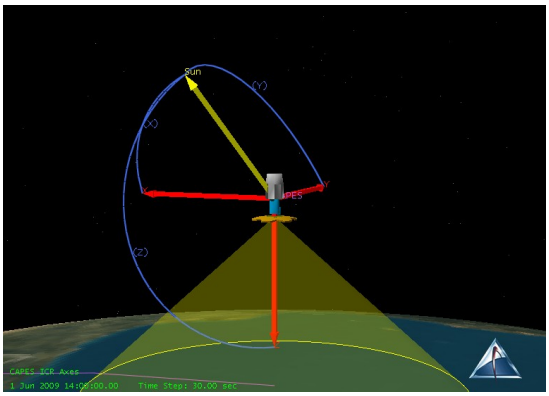
# Integrated Design Center
# Operating Manual

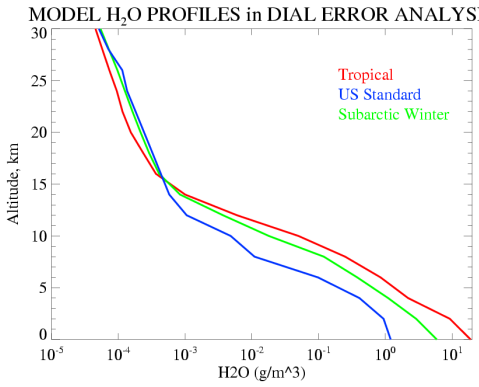
The IDC is intended to allow improved linking of all software tools, as well as collaboration and knowledge sharing among team members.

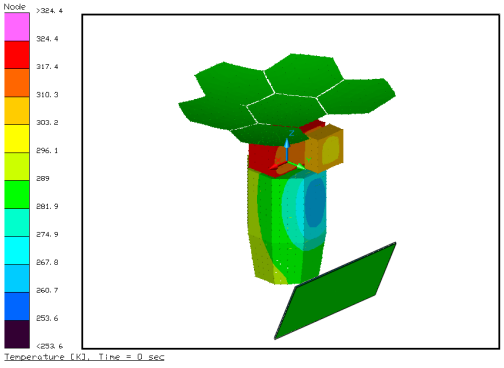
# Volume 2: Detailed Discipline Interfaces




7/14/2011

# Langley Integrated Design Center (IDC) Procedures: Discipline Processes and Interfaces between Stations

*Note: This is a very large document. This best way to get where you want to go is look at the Table of Contents, and Ctrl-click to get to the desired section.*

*This is a living document. Please send recommended changes to*

**Ruth.M.Amundsen@nasa.gov**
**Danette.Allen@nasa.gov**

## Table of Contents

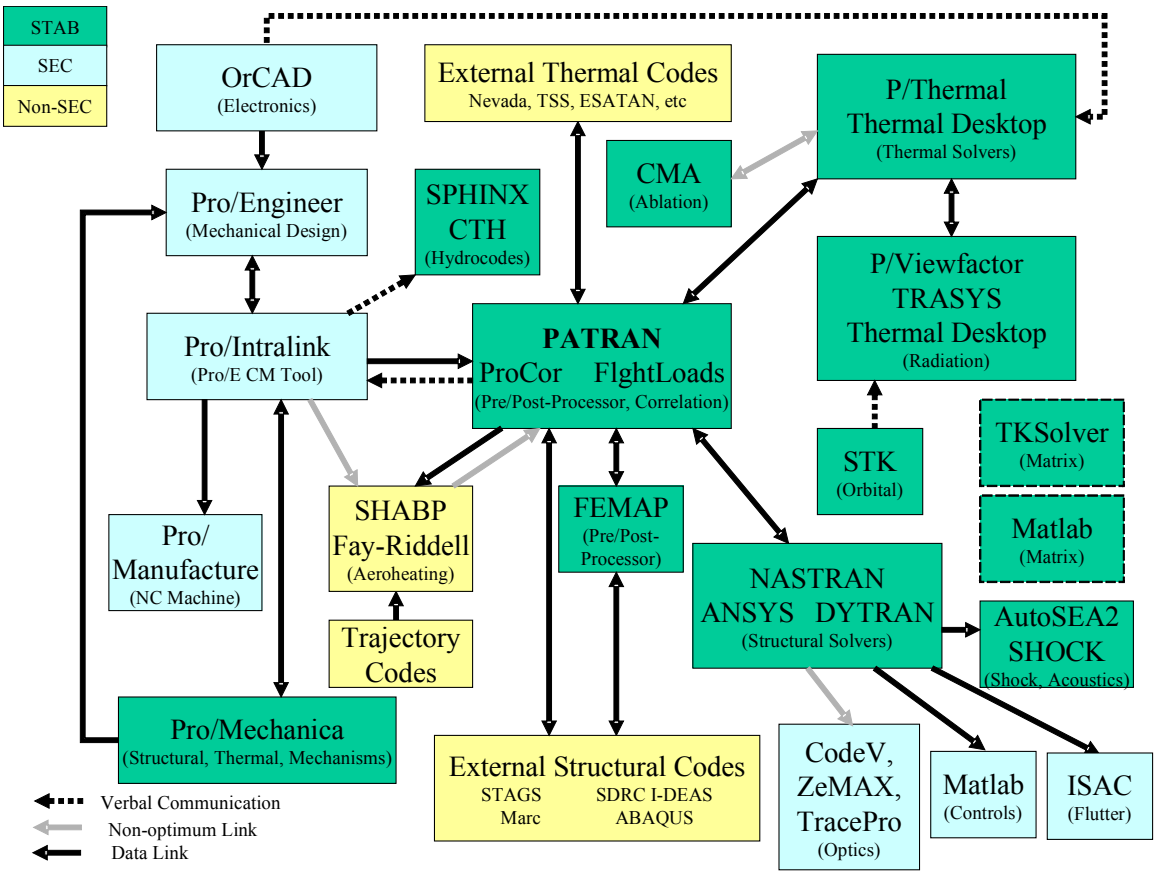
# Tools

Table 1 shows an example of the software on each station for the initial pilot run in the IDC.

**Table 1. LaRC IDC Disciplines and Software Tools**

| IDC Station | Tools |
|---|---|
| Software | MS Visio |
| | MS Visual Studio |
| Structural | PATRAN |
| | PATRAN Thermal |
| | NASTRAN |
| | SigFit |
| | PATRAN Materials Selector |
| Mech/CAD | Pro/E |
| Active Payload (Lasers) | STK |
| | ZeMAX |
| Passive Payload (Optic System, Detectors) | ZeMAX |
| | TracePRO |
| | STK |
| Avionics/Electronics | |
| Orbital/Environmental | STK |
| | PolyTrans |
| Telecom | Multi Mission Telecom Analysis Tool – MMTAT |
| | MatLab |
| | STK |
| | SPASIM |
| Thermal | PATRAN |
| | PATRAN Thermal |
| | Thermal Desktop / AutoCAD |
| | Fortran |
| | C++ |
| Fabrication/ Testing | Pro/E |
| | Unigraphics |
| Optics | ZeMAX |
| | TracePRO |
| Mission/System, Customer/Science, Documentation | |
| Power | OrCAD via Windows 2000 terminal emulator |
| | JPL S/W -- MMPAT |
| All | Remote Admin, plus Standard ODIN load: MS Office suite, etc. |

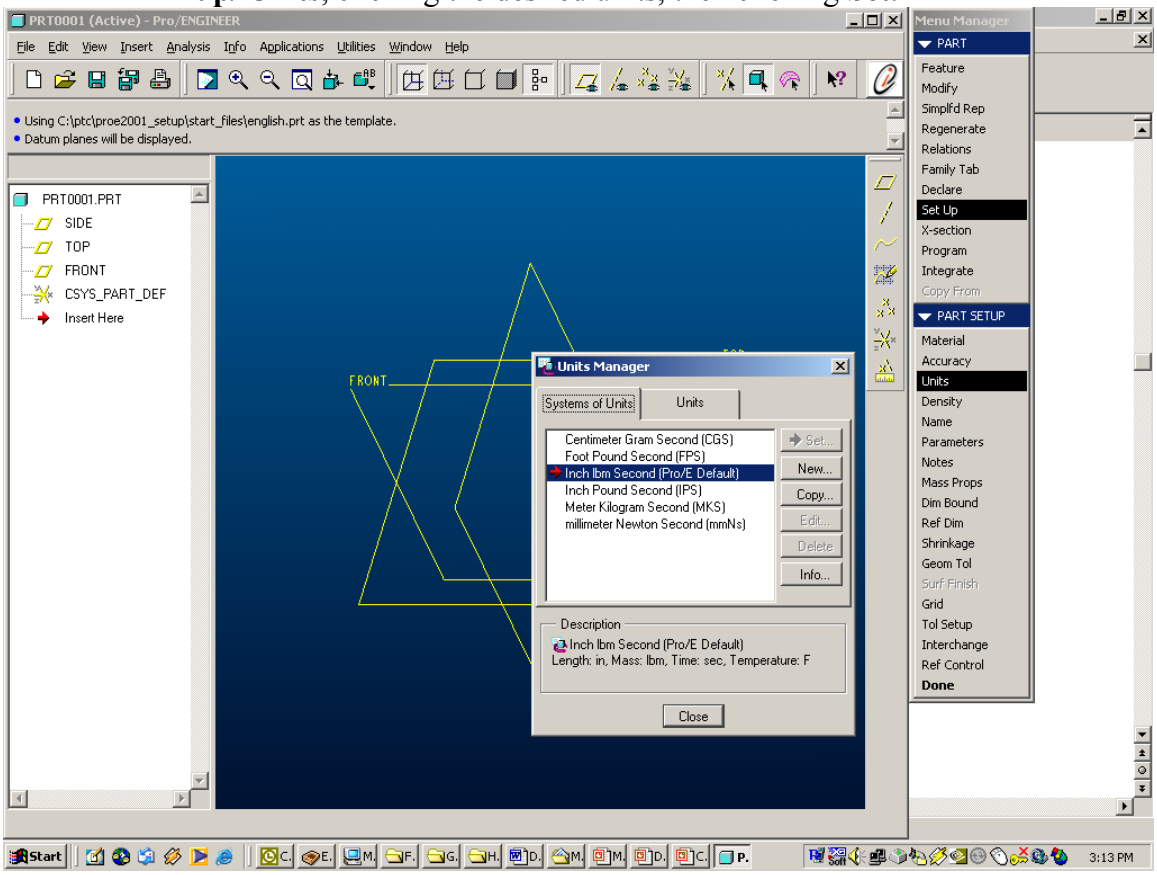
Current software load is listed in “IDC Workstations.xls” on the IDC shared drive.
Shown below is a partial diagram of how some of these tools interlink.


STAB
SEC
Non-SEC
OrCAD
(Electronics)
External Thermal Codes
Nevada, TSS, ESATAN, etc
P/Thermal
Thermal Desktop
(Thermal Solvers)
CMA
(Ablation)
Pro/Engineer
(Mechanical Design)
SPHINX
CTH
(Hydrocodes)
P/Viewfactor
TRASYS
Thermal Desktop
(Radiation)
Pro/Intralink
(Pro/E CM Tool)
PATRAN
ProCor FlghtLoads
(Pre/Post-Processor, Correlation)
TKSolver
(Matrix)
STK
(Orbital)
Matlab
(Matrix)
Pro/
Manufacture
(NC Machine)
SHABP
Fay-Riddell
(Aeroheating)
FEMAP
(Pre/Post-
Processor)
NASTRAN
ANSYS DYTRAN
(Structural Solvers)
AutoSEA2
SHOCK
(Shock, Acoustics)
Trajectory
Codes
Pro/Mechanica
(Structural, Thermal, Mechanisms)
External Structural Codes
STAGS SDRC I-DEAS
Marc ABAQUS
CodeV,
ZeMAX,
TracePro
(Optics)
Matlab
(Controls)
ISAC
(Flutter)
Verbal Communication
Non-optimum Link
Data Link

# Mechanical Station: CAD Practices for IDC

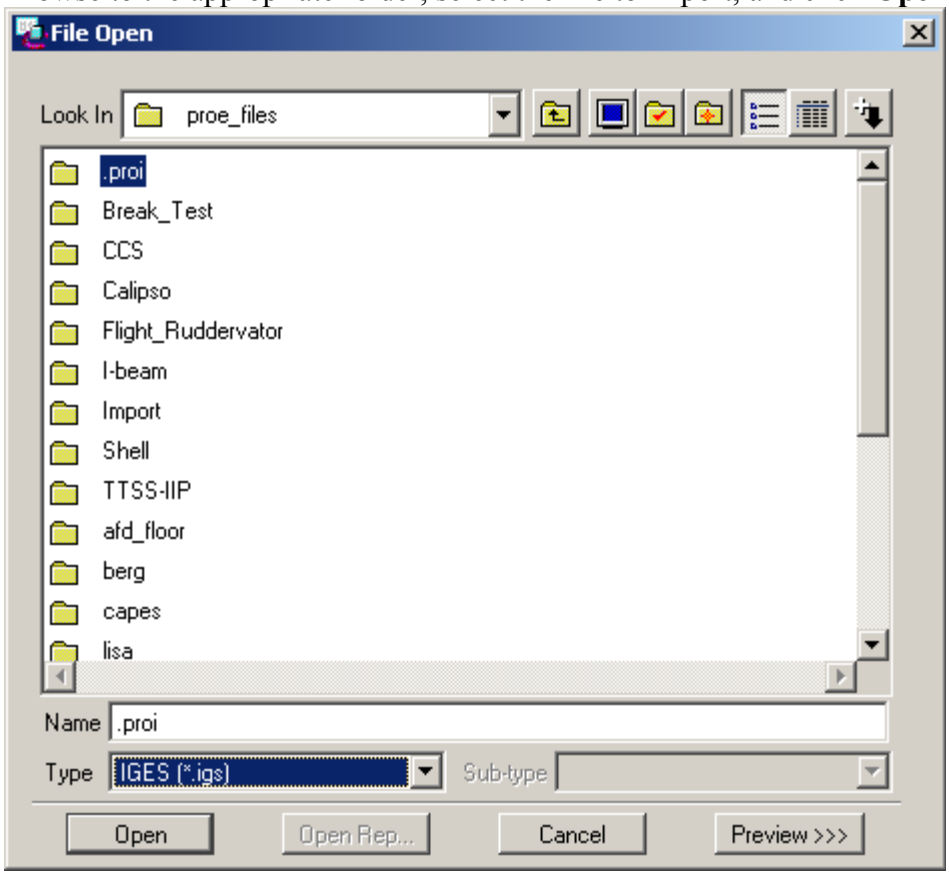
This document is an overview of best practices to be used by a mechanical designer in a Integrated Design Center session. Pro Engineer is assumed to be the primary CAD tool used by the mechanical designer. This is a working document and will evolve as software is upgraded and new software and translators become available. Included below is practices on creating geometry, Importing and exporting data, and interfaces between mechanical design and other disciplines involve in the design process.

1. Creating Geometry
    a. Coordinate System – Before any CAD modeling is done, agreement on orientation should be made and documented at the project level. The model should be built in an orientation such that the project coordinated system is the default CAD coordinate system. Although coordinates system can be added at any time, exporting and importing assemblies is more stable using the default coordinate system.
    b. Units – Units should also be set at the project level. Units are set by clicking **Set up/ Units**, clicking the desired units, then clicking **Set.**

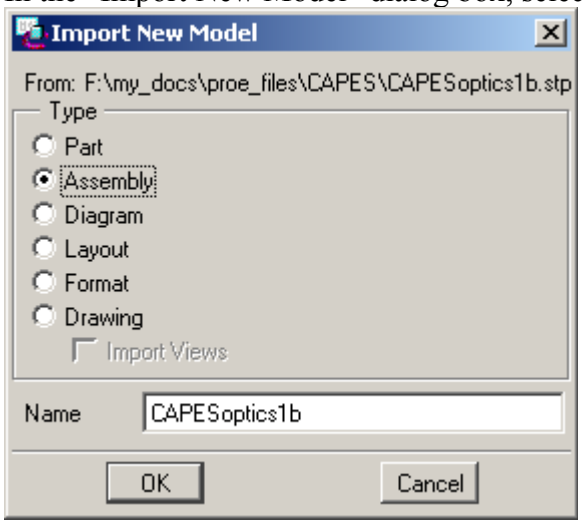



    c. Start Parts – Start Parts and Assemblies should be created for each project. Models created using start parts will have the same coordinate system and consistent units as well as share a similar layer scheme.
    d. Large or complicated assemblies should have skeletons.
2. Importing

a. Importing 3D geometry
   i. From the **File** pull-down menu select **Open**.
   ii. The "File Open" dialog box will appear. Set the "Type" (i.e. IGES, STEP, SAT…).
   iii. Browse to the appropriate folder, select the file to import, and click **Open.**




   iv. In the "Import New Model" dialog box, select **Assembly**.

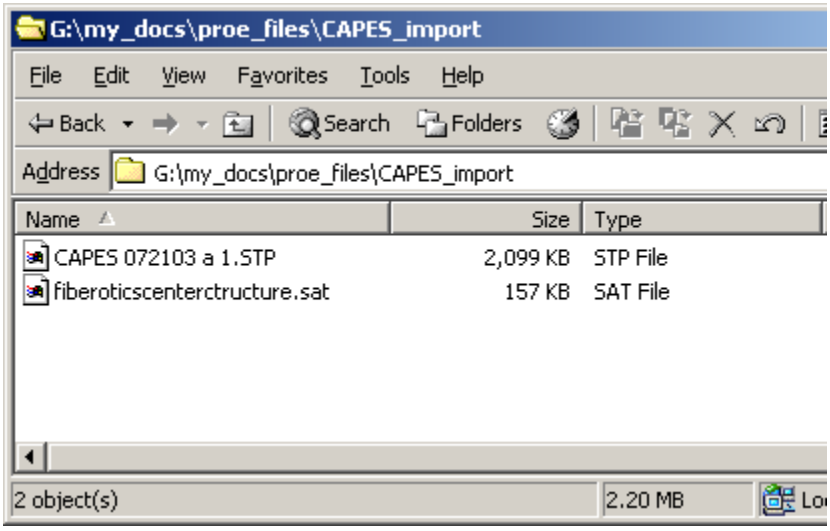



   v. Rename the model if desire then click **OK.**

b. Importing 2D data can be done in a similar process to 3D data.
   i. From the **File** pull-down menu select **Open.**

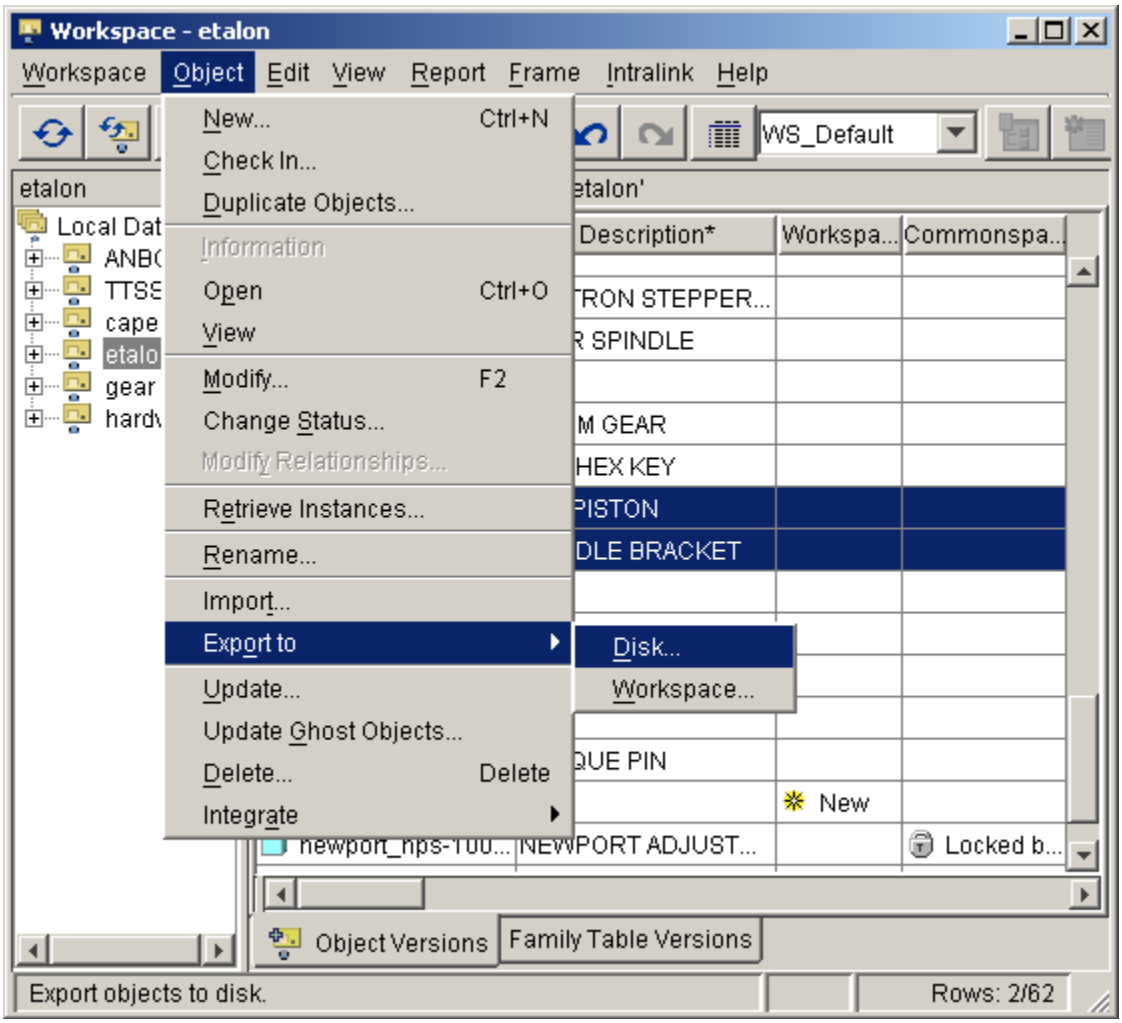
ii. Select **Drawing** as "Type."
iii. Enter **No** to "Scale to fit format?"
iv. Enter **Yes** to "Move bottom left corner of drawing to screen origin?"

c. Troubleshooting – If the model fails to import, check the following:
   i. ProE and DOS are more flexible with file name than the translators ProE uses. Filenames can be made up of letters, numbers, dashes, and underscores. Spaces should be removed. Also, the files extension is case sensitive. In the figure below, "CAPES 072103 a 1.STP" should be renamed to "capes_072103_a_1.stp." Although it is not required to change CAPES to lower case on a windows platform, it is preferred sense ProE on unix is case sensitive.

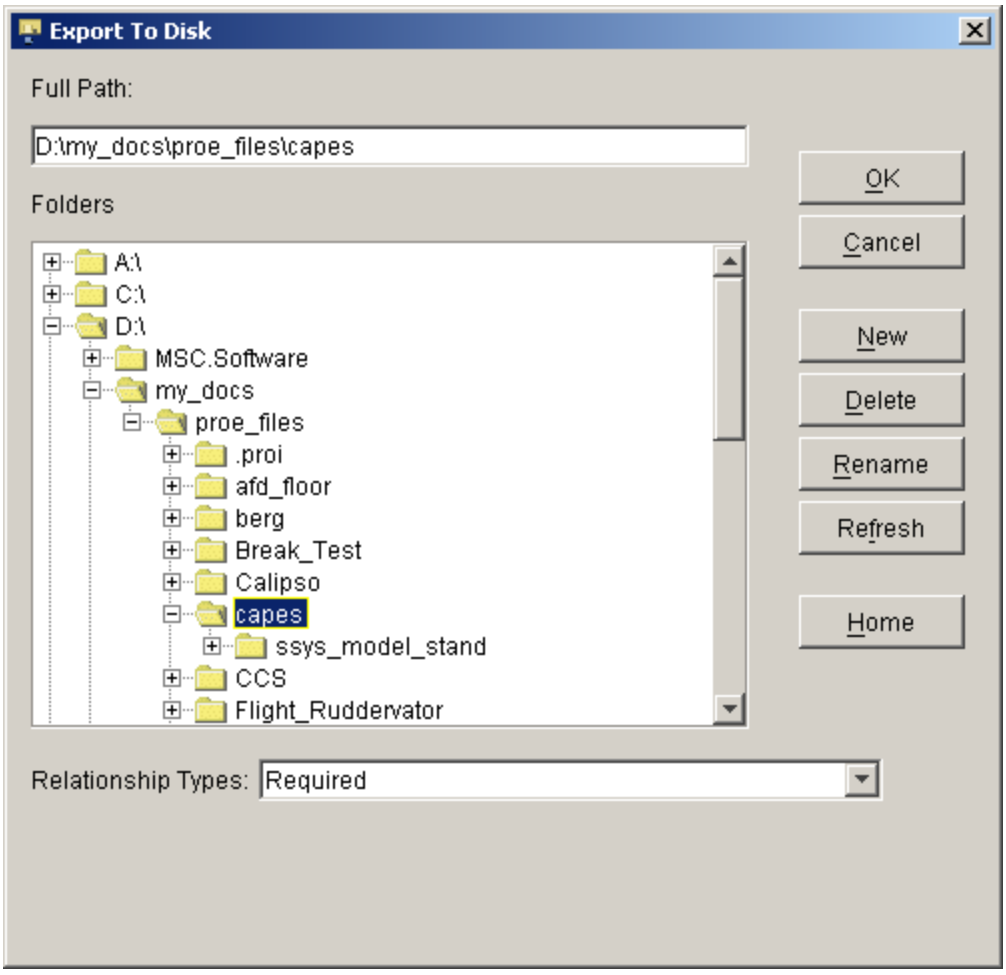



   ii. Check the file size of the file you are trying to import. If it is a zero length file or the file is larger than you would expect, there may have been an error creating the file.
   iii. Some translators are version specific. Check with software manufacturers to assure what versions will translate.
   iv. Surface geometry may require some repair after translation. Suggested technique for fixing surfaces can be found at ptc.com.

3. Exporting
   a. Exporting ProE files – Several CAD applications have embedded translators that allow the software to read ProE data directly. When using Intralink to manage the ProE data, use the following steps to make the files available for use by another program:
      i. Save the current session of ProE.
      ii. Open a Workspace Browser.
      iii. Select the files you wish to export and click **Export to** then **Disk…** from the **Object** pull down menu.




iv. From the "Export to Disk" dialog box select **Required** for the "Relationship Types:" and browse to select the target folder.

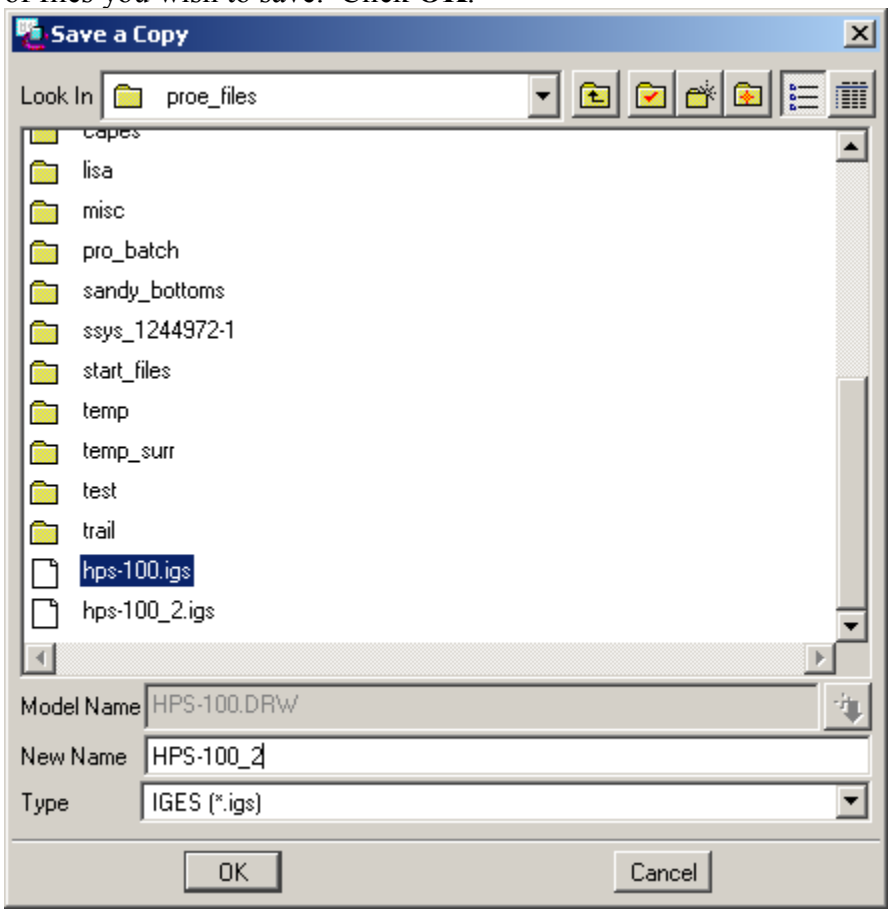



v. Click **OK.**

b. Exporting Using Translators

i. To export using translators click "**Save a Copy…**" from the **File** pull down menu. The "Save a Copy" dialog box will appear.

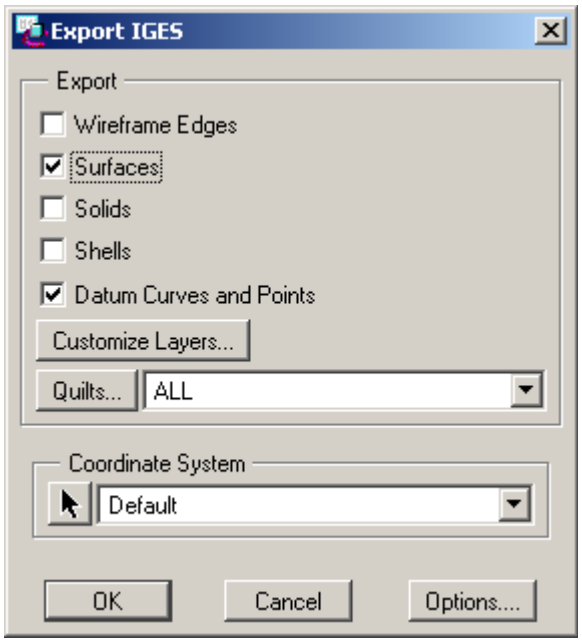
ii. Enter the “New Name” for the model to be exported and select the “Type” of files you wish to save. Click **OK**.




iii. The “Export IGES” dialog box will appear. Select the feature types you wish to export.

iv. Use the default coordinate system whenever possible.

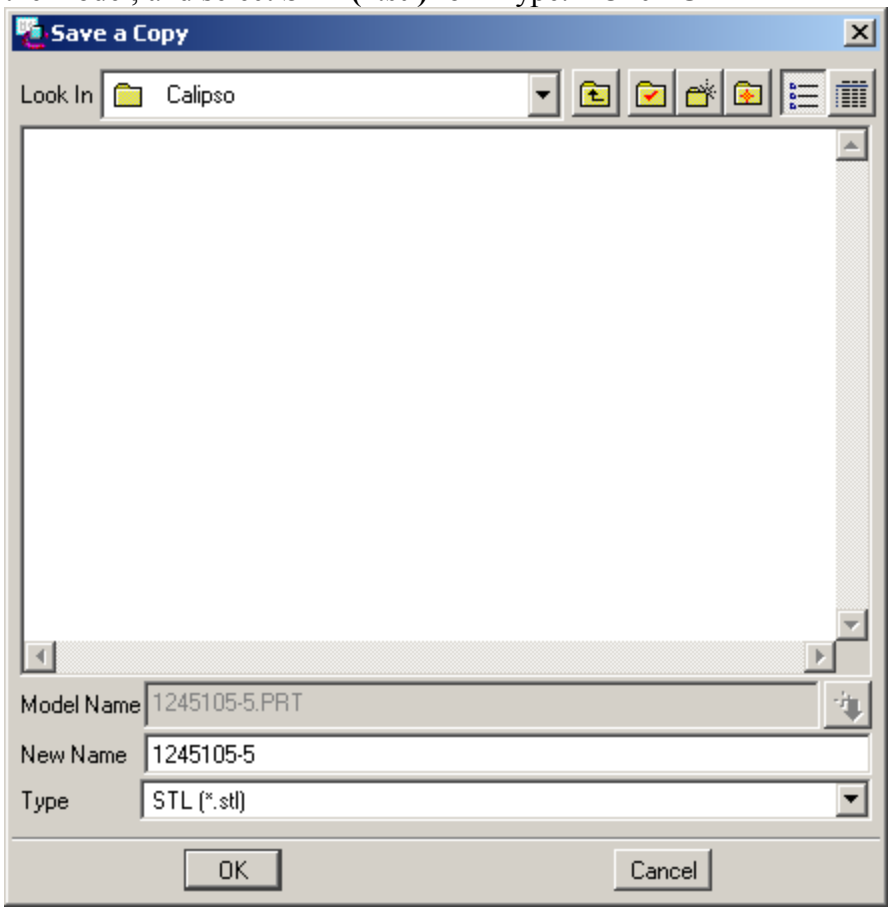
v. Click **OK.**



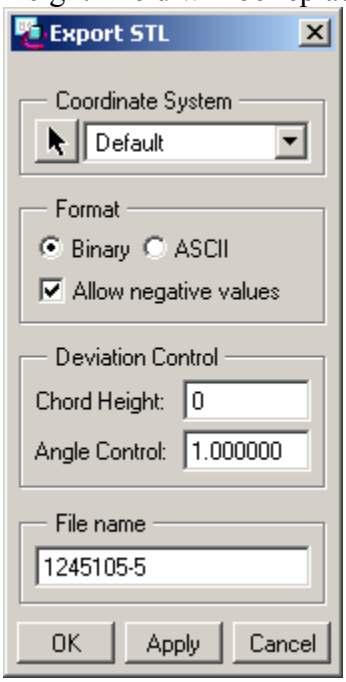

c. Exporting to STL

i. From the **File** pull down menu click **Save as Copy…**

ii. The "Save as Copy" dialog box will appear. Enter the "New Name" for the model, and select **STL (*.stl)** for "Type." Click **OK**



iii. When the "Export STL" dialog box appears, enter **0** for the "Chord Height," **1** for the "Angle Control," and click **OK.** (Note: The "Chord Height" field will be replace by the minimum acceptable value.)




4. Interfacing with other stations – Importing and exporting data between the different disciplines of a design team often requires a third party or intermediate translator. The following is the interfaces between Mechanical CAD and other stations used in the development of the CAPES pilot.
    a. Optical – Geometry was imported into ProE using Asis(.sat) and IGES (.igs) formats. IGES was used for curves and optical rays.
    b. Structural – Geometry, mass, and CG information from ProE to Patran using translators imbedded in Patran.
    c. Thermal – The primary path of information from mechanical design to thermal is through Structural. Some geometry was transferred from ProE to Thermal Desktop using STEP and IGES translators.
    d. Orbital – Geometry and mass information was exported from ProE to STK using IGES as a translator.
    e. Fabrication
        i. For NC machining, 3D Geometry can be imported directly into Unigraphics from ProE using an embedded translator.
        ii. For plastic prototyping or casting, a 3D model can be exported using STL format.
        iii. For water-saw cutting, silk-screening, and other 2D processes, geometry can be exported from ProE using DXF and IGES translators.
        iv. All information from Engineering to Fabrications will be documented on a detail drawing. Detail Drawings will include the following information:
            1. Geometry definition – if part is not dimensioned of drawing the a note for the electronic file will be included.
            2. Materials definition

3. Heat or chemical treatment
4. Tolerance information
5. Inspections and Special Operations will be conveyed on a FIOS (LaRC form 136)

# Thermal Station – Best Practices

## *Guidelines for selection of thermal analysis tool*

These are general guidelines. There will always be cases where a different method will be used for reasons of experience, previous history, collaboration with other entities, etc. However, these are rules to consider when selecting a tool at the initiation of a modeling effort. Many analyses will be a combination of two or more of these analysis types, so several tools may be used.

| Analysis characterization | Tool Recommendation | Comments |
|---|---|---|
| Part or assembly geometry exists in Pro/Engineer. No orbital influence. No complex fluid flow. No MLI. | PATRAN Thermal | Import geo directly from Pro/E |
| Problem largely influenced by complex fluid flow, speed less than Mach 5. | CFDesign | Import geo directly from Pro/E |
| Problem largely influenced by orbital (solar, planetary and space) radiation. | Thermal Desktop / RADCAD | Import geo as STEP or IGES from Pro/E. If suitable NASTRAN mesh exists, import that instead. |
| Problem includes large amount of MLI that is the principal influence on performance. | Thermal Desktop / RADCAD | MLI application, modeling and visualization is much easier in Thermal Desktop than in other tools here. |
| Problem largely influenced by aeroheating of hypersonic flow. | PATRAN Thermal | Currently, aeroheating interpolation code exists in moderately robust form in the ulib for PATRAN Thermal. However, it is possible that we will be able to incorporate this functionality in Thermal Desktop in the next several years. |
| Problem largely influenced by ablation. | PATRAN Thermal | Currently, we have modified Patran to incorporate ablation using CMA within PATRAN Thermal; however, display is not functional. A simplistic ablation capability exists within Thermal Desktop (ABLATE). |
| Ultimate performance is dependent on thermal stresses. | PATRAN Thermal or Thermal Desktop | Transfer of thermal map to NATRAN mesh is simple and well-tried in both these tools (unknown if this is smooth CFDesign). |

| Analysis characterization | Tool Recommendation | Comments |
|---|---|---|
| Simple model with less than 10 nodes is all that is needed. | SINDA, TAK2000, Thermal Desktop | Manual model building is clumsy and can be error-prone for models larger than 10 nodes. |

## *Short Guide to Thermal Desktop*

In Patran you usually start out by following the order of radio buttons, geometry, elements, Loads/BC's etc, although you can go in any order you like. You can do the same in Thermal Desktop, although it may be easier to go in the following order just because of the way TD is set up:

TIP: If you are looking for something specific, use the search feature in Word to find a term.

1. **Defining Materials and Radiation Groups**
2. **Build Geometry using thermal surfaces, or simple solids** (I would make sure you make lots of layers for each type of thing your including in your model. Its hard to select "types" of objects in TD individually. Complex solids are difficult to do in TD. If you have the time, you can create individual nodes and elements in 2-D and the extrude/revolve or whatever into 3-D elements. If you do this make sure you check the "delete planar elements" box to get rid of the 2-D surfaces which may cause problems when you try to solve. You can also create AutoCAD Surfaces and then convert them into FE's. This is explained in the creating Geometry section.)
    a. **Create any Boundary Conditions or Loads** (You do not need to create radiation BC's as will be explained. Radiation is automatically taken care of from any object that has been assigned an optical property. I do not know if there is a convection option or not, however you can make fluid flow networks that absorb and transmit heat.)
3. **Model Checks**
4. **Set up solution control and run solution**
5. **Post Processing**
6. **MISC. Features of AutoCAD that are Useful to Know**
    a. Viewing, Geometry size, and other Prefs.
    b. Layers
    c. Modify Menu
    d. Model Infrared or Solar Heat Lamps

The rest of this document will go through the above steps in detail.

### 1. DEFINING MATERIALS AND RADIATION GROUPS

You may want to set up a template file to save your viewing preferences, whatever they may be. I am not sure if you can set defaults or not, but every time you make a new file all viewing settings will be reset to TD's defaults.

Define your Materials:

1. *Thermal – Thermophysical Properties – Edit Property Data*. A dialog box will appear. This is where you create your materials. Type a name for the material in the "New Property to Add" box (unlike Patran you may use spaces, so don't worry about underscores). Click add (note: do not press enter, this will select the ok button, the dialog box will close, and the material will not be created). Fill in the properties of the material (note: double clicking in a text box will bring up the symbol manager form, it will not highlight the number like normal programs, which is aggravating). You may enter temperature dependent properties if you want by clicking the "edit table" button next to each of the input boxes (you also must select the "Use Table" check box next to the button). There is not a spreadsheet in this form, just enter your points separated by a comma and press enter, i.e. 0,3.5 "enter" 10,4.5, etc. A graph will appear on the right hand side to give you a visual aide of what you are inputting which is nice. You may also select isotropic or anisotropic in this dialog box.
2. If there will be radiation in your model, *Thermal – Optical Properties – Edit Property Data*. In the "New Property To Add" box, enter a name for your radiation property, you may enter the same name you did for a thermophysical property, so if you wanted to call both steel you can. Otherwise you may call it some kind of paint, or whatever it will be. Click the add button. This will again bring up properties for radiation, and again you may put temperature dependent properties by clicking the edit table button next to each text box.
3. You may also want to create multiple radiation groups. This will allow you to have some objects that will not see radiation from another object, or will not radiate to another object. You can create different groups by using the *Thermal – Radiation Analysis Groups…* command. A detailed description of how multiple groups are used can be found in the Lamps section of this report.

## 2. BUILD YOUR MODEL

There are 2 ways to create your models: If you are making simple 2-D objects, like using shell elements in Patran, you can create thermal surfaces. If you wish to create complex 3-D solids, you will need to use finite elements. Both methods are described below

Using Thermal Surfaces

1. Select the type of surface you wish to create from the toolbar, or from *Thermal – Surfaces/Solids*.
2. TD surfaces have kind of a strange input for creating them in your workspace. Each object has its own coordinate frame, and you must align this frame with the global frame to create the object. The following steps show how to create a square or rectangle:
    a. Pick the thermal rectangle icon on the toolbar
    b. The first prompt on the command line is "Origin Point" which defines the first corner of the rectangle, pick one from the screen or enter it. You may supply only x and y coordinates if you wish, you do not need to supply a z unless necessary. What is shown in the < > brackets are the default; leaving the command line blank and pressing enter will use this value. Press enter when you have entered your point.

c. The next prompt is "Point for +x Axis and x-size". This asks you to define the x axis *of the rectangle* with respect to the global coordinate system. By default, the x axis is along an edge of the rectangle, the y axis is along the other edge, and the z axis is normal to the surface. If you wish the rectangle to be in the x-y plane, and its x axis to line up with the global x axis, and your origin is at 0,0,0, you pick a point on this line, i.e. 2,0,0. If you wish to use a displacement from your origin point, simply put an @ in front of your point, so @2,0,0 will set the second corner 2 in the positive global x direction away from the origin point, and none in the other directions. Press Enter.
d. The third prompt is "Point to set XY plane and Y-size". Here you enter a point for the third and final point of the rectangle. Again you can use the @ sign for a displacement from the second point. Press Enter
e. When you press enter a dialog box will appear. You should go through each tab to set up the thermal properties:
    i. Subdivision. This is where you enter the amount of nodes in your rectangle (or other thermal object). Centered nodes will put the nodes in the center of the elements. Edge nodes will put them at the edges. You must use Edge nodes if you are planning on having conduction between different surfaces (use centered nodes if you are planning on having a contact resistance between 2 surfaces. In this way you can assure that the nodes do not get merged). You then enter the the number of nodes you want in each direction. I'm not sure what the "list" option does
    ii. Numbering. Here is where you set the node ID's. If you choose a start number, it will start numbering nodes at that number in increments of 1. Be careful, by default the start number is always 1 whenever you create a surface. This is aggravating because you will have the same node numbers in different surfaces unless you manually change the start number to be one more than the last node number in your previous surface. If you don't care bout specific node number this is ok, you can just renumber nodes before you run your analysis, or any time you like (explained later). Sometimes you want a surface to contain nodes of all the same number (when making heat lamps for example). The only way I know to do this is to use the list option instead of the start ID option, and input a list of the same number for however many nodes are in your surface. If there are a lot of them, you may want to create a list in Excel or Word and paste it in. You can have the back and front sides of the nodes labeled differently if you wish by unchecking the "use same ID's on both sides" box (possibly useful for radiation models, however I don't know how to coat each side individually with an optical property).
    iii. Radiation. In this tab you set the radiation properties of the surface. First you need to assign this surface to specific radiation groups, this means basically that the object will be able to see radiation from these groups. By default, there is only one available model called BASE. If you wanted to have separate groups so that certain objects do no radiate to each other you can do this by *Thermal – Radiation Analysis Groups…* Here you can create new groups or copy current groups (see the Lamps section of this

report for a clearer understanding of this). Double click on the Radation Group you would like to add the object to. A dialog box appears allowing you to select in which directions the surface will radiate. "Top/Out" will set radiation in the positive z direction, "Bottom/In" will set it in the negative z direction, Both does both, none does none, and "not in analysis group" is used if you do not want this object to see radiation from this group. Once you have selected your radiating direction, hit ok, and next select what optical property you want each of the sides to have. If you leave it as "Default" Thermal Desktop takes it to be a black body.

iv. Cond/Cap. In this tab you set your thermo-physical properties. Just keep the "generate nodes and conductors" checked, and make sure you are on "main" and "based on material property". There is also an Arithmetic node option if you want zero capacitance. In the material drop down list select the material that you want. The thickness text box is used to input your surfaces thickness (same as shell thickness in Patran for 2-D shell elements). In some cases you may not want conduction to take place in a body; something that you don't care what its temperature distribution is, only that it radiates or absorbs radiation. In this case you can uncheck the "generate nodes and conductors" box.

v. Contact. This tab is used to make contact resistances on your surface. You can make them on the top or bottom of the surface, or along an edge. Remember that for an edge, the x and y coordinates are for the *surface* x and y coordinates, which may or may not correspond to the global coordinates.

vi. Insulation. Can be used to put insulation on your surface. This is for thermophysical insulation, not optical (I don't think, there is nowhere to put any optical properties on it, maybe it just takes the optical properties of the surface)

vii. Surface. You can resize the surface if you wish

viii. Trans/Rot. You can translate or rotate the surface if you wish.

(NOTE: You can get back to this dialog box at any time. Select the object you wish to edit, and click on the "edit thermal properties" Icon, or use *Thermal – Edit…* sometimes you will select more than one type of object (nodes and surfaces for example), in this case a filter box will appear; select which type of object you would like to edit (nodes or surfaces))

f. The preceding steps are similar for all thermal surfaces, they are created by setting up their coord frame with respect to the global one, and then editing its thermal properties in the dialog box that appears. (this is why it is easier to input all your materials before you start drawing anything so that you don't have to close the dialog box, create a material, and then use the edit command to select that material)

3. Renumber the nodes so that they are all individual using *Thermal – Modeling Tools – Resequence ID's*. If you do not do this step, it should be ok, when you run the analysis it will ask you if you want to resequence ID's if it finds more than one with the same

number. Some times you might want a surface to have nodes of the same number (see Numbering). You will also need to use the *Thermal – FD/FEM Network – Merge Coincident Nodes* command if you have surfaces with edge nodes that will be conducting to each other.

(NOTE: If you will be placing thermal surfaces next to each other that will be conducting to each other, it is a good idea to make sure the nodes line up. If the do not, the nodes will not merge when you use the merge command, and possibly only one node may be merged REMINDER: you must use edge nodes for surfaces that will be conducting to each other with no contact resistance)

Creating FE meshes

It is easier to create a 2-D mesh in Patran and import it into AutoCAD. But if you want to do it in TD:

1. Draw complex 2-D shape that you want using regular AutoCAD geometry commands, shape must be broken down into shapes with 4 sides, and these sides must be individual lines, but they do not have to be straight lines. Pretty sure surfaces must be congruent, if they are not then you must supply equal x and y number of nodes in the next few steps.
2. Type surftab1 and surftab2 into the command line to change the number of nodes in each direction (explained in 4.)
3. Use "edgesuf" and pick all four sides of a closed surface you would like to mesh. The first side you pick will have the number of nodes inputted into surftab1, and the second side you pick will have the number of nodes inputted into surftab2.
4. In order to select the same side for a new block, you need to create a layer with only mesh, make it active, and turn it off. This will put all the created mesh into this layer but they will not be visible so that they don't cover up the geometry. If you do not do this you can't select geometry that has mesh on it. (probably another way but I'm not good with AutoCAD commands, see Layers for help with layers)
5. Once all the grids are done use the *Thermal – FD/FEM Network – Convert AutoCAD Surface To Node Element* and select your entire model. Press enter and the standard thermal properties form will appear. This has created a FE nodal network like Patran's mesh command does.
6. Merge coincident nodes using *Thermal – FD/FEM Network – Merge Coincident Nodes*.
7. Once a 2-D shape has been created in this manner, it may be extruded to 3-D if needed using one of the "Extruded" options in the *Thermal – FD/FEM Network* menu.
8. Renumber the nodes so that they are all individual using *Thermal – Modeling Tools – Resequence ID's*. If you do not do this step, it should be ok, when you run the analysis it will ask you if you want to resequence ID's if it finds more than one with the same number. Some times you might want a surface to have nodes of the same number (see Numbering).

Note: This process takes quite some time to complete. You must remember how many nodes you put on each side of each 4 edge surface you mesh. It is also a pain to keep turning the active mesh layer off and on to see what you've drawn. To make it easier you can import mesh from Patran.

REMEMBER: Once you have created your geometry using either method, you should always use the merge coincident nodes command, and the resequence ID's command as described above. It is not critical to renumber ID's because it will ask you if you want to do this when you run the analysis, but it is a good idea if you have a surface that has a multiple nodes with the same number that you would like to keep that way. The resequence command lets you select only the surfaces you want to resequence, while the option during the analysis run will resequence all nodes.

## 3. **BOUNDARY CONDITIONS AND LOADS**

This section will describe how to add BC's and Loads to your model one by one.

Fixed temperature condition

1. Select the nodes whose temperatures you wish to fix.
2. Click the Edit thermal properties icon or use Edit under the thermal menu.
3. Check the "Override Calculations by surfaces and Elements" check box.
4. The shadowed options will become available. Select "Boundary Node" and then input the temperature you want into the "Initial Temperature" text box. This will keep the nodes fixed at that temperature throughout the analysis.

Heat Loads

1. *Thermal – FD/FEM Nodal Network – Heat loads on …* where you can select *on surfaces*, *on nodes*, or *on solids*.
2. Select whatever it was you wanted to apply the load to.
3. A dialog box pops up, enter the value for the heat load, and choose whether you want it to be a flux, or absolute. You can only use flux on a surface as there is no area associated with a node.
4. Click OK. Red arrows will appear indicating the direction of the load and what it has been applied to.

Conduction

Just a reminder here; you must make sure that any surfaces you wish to conduct between are built with edge nodes. Also make sure you use the *Thermal – FD/FEM Network – Merge Coincident Nodes* command before running your analysis, other wise surfaces will not be connected. You should verify that your element boundaries before starting a solution (see next section, Model Checks)

Convection

In order to create convection, you must go through the following procedure. Depending on how many convection connections you have, the solution procedure can be very slow. Convection seems to slow down the solution more than any other type of load or BC.

1. You must first create a fluid lump, this is the same as creating a fixed temperature node in patran for your convection condition. You can do this using the Create Lump icon on the far left toolbar, or by *Thermal – Fluid Modeling – Lump*.
2. Next, you must tie the lump to either the nodes or surfaces you want to convect to. There are icons underneath the create lump icon to tie to one of these two items. Or *Thermal – Fluid Modeling – Tie to node/tie to surface*.
3. Next you must edit the lump. Select it and click the thermal edit icon. Enter the fixed temperature you would like the fluid to posses. In the "Type" section, select the "Plenum (Boundary Infinite Volume)" radio button. There are other options here but I'm not sure what they are for. You can not complete this step before completing step 2. If you try to do so, AutoCAD will say "no lumps selected" when you try to select the plenum lump.
4. Next click on the nodal ties that are visible, and click the thermal edit button. The "Tie Generation Direction" should be set to "node to lumps". Enter your heat transfer coefficient in the text box; you can then choose absolute or per area. There are other options, not sure what they do.

Radiation

Radiation is not controlled by adding a "condition". Everything that you gave an optical property and a radiating direction will radiate to everything in your model including space (as long as everything is in the same radiation analysis group). You do have to make sure, however, that the Active Sides of the objects are facing in the direction that you want them to radiate. If they are not, they may be radiating the wrong way (see next section, Model Checks). There also may be cases where the normals of elements of the same surface are not in the same direction (not likely unless you delete a particular element then redraw it, or draw all your nodes and then create elements manually). If this happens, you can use *Thermal – FD/FEM Network – Synchronize Element Normals*. Click on the element that faces in the direction you want all the elements to face. Next select all the elements you want to face that way. Press enter.

## 4. MODEL CHECKS

There are a number of options for checking your grid and surfaces, I only use a few that may be helpful:

1. Show Active Surfaces – This is kinda the same as showing surface normals in Patran. You can either use coloring or arrows. When you want a surface to radiate to another, you must make sure that their active faces are pointing toward each other, it may be easier to see with arrows. To do this, use *Thermal – Model Checks – Active sides Preferences*... This will allow you to choose either color code, or arrows. Next choose *Thermal – Model Checks – display active sides.* This will show the arrows or colors. *Turn off Active sides* will hide them.
2. Show Free Boundaries. This is the same verify/element/boundaries in Patran and will show you if you have any connectivity issues. This is under *Thermal – Model Checks –*

*Show Free Edges*. You will have to select the surfaces you want, or you can type "all" in the command line to select everything. The free edges will be highlighted in red.

3. There are many other useful options under the *Model Checks* menu but I have not yet experimented with them.

## 5. SETTING UP AND RUNNING AN ANALYSIS

1. *Thermal – Case Set Manager…* This will bring up the Case Manager, which is the same as the Analysis radio button in Patran.
2. The default case is Case 0. You can add as many cases as you like by clicking the Add button, each one will contain a specific solution, just like being able to specify different names for different analyses in Patran. You can also specify the names of the files that will be outputted.
3. Once you have created your case, highlight the one you want to run in the list and select the Properties button. Go through all the Tabs:
    a. Radiation Tasks. Specify whether you want to include radiation in your model. If you do, select the radiation group you want to make active, click the Add button, and then the Properties button. Go through the tabs in the dialog box that pops up:
        i. Control tab allows you to specify how many rays you will be shooting as well as options that may speed up calculations. You can also specify which nodes in the radiation group to include.
        ii. Advanced Control tab gives you more options for speeding up calculations, and lamp modeling options.
        iii. Radk Output tab gives lets you rename the radiation output file names, and lets you change the space node temperature. There is an option for outputting radiation as heat rates which is used for modeling lamps.
        iv. Spin tab, no idea what this does.
    b. On the right side of the Radiation Tasks tab are options for the analysis. If rays have already been calculated, you can use those for the analysis, create new ones, or create new ones and include ones already created.
    c. S/F Calculations. Here you set all your solution type properties. You can rename the SINDA output files if you wish. You want to have all the check boxes checked on the left hand side. Pick either transient or steady state (if you are running steady state, make sure that you have 0 in both the start time and end time of the transient options, if you don't, there will be a warning after your solution is completed; it doesn't affect anything, it is just annoying trying to figure out where it is coming from). In the Convergence Criteria section, you will want to increase the Max iterations for steady state runs. If you don't the routine will terminate after whatever number is there even if it has not converged (it will tell you if this is the case). For transient, 1000 seems to work ok, although a greater number may be more accurate (solution time will increase greatly with higher numbers). The Max Temperature Change and System Energy Level Balance boxes should be reduced for complicated models, especially steady state runs. If you don't

reduce them, your steady state solutions may not really be steady state, more energy will be entering then exiting (it will give you a warning if this is the case).

d. S/F Output. Control what you want to be outputted. Select an output increment for time. You will only be able to view the solution at this time step interval for transient runs. If you are running steady state, you will want to put 0 here. You can change the name of the files that will be outputted, and below that, you can change what data will be written in them.

e. All Other Tabs. No clue

4. Select OK on this dialog box. Check the "save drawing before running" box, and set the Duplicate Nodes drop down box to "prompt if found". This will allow you to renumber the nodes if you have not already done so (in some cases you may not want to renumber duplicate nodes; if this is the case, you can set this drop down to allow in model, or simply click the Allow button when it notifies you).
5. Click Run case. The solution will proceed, and when it is finished it will bring you directly into post processing mode at the first time step.

(NOTE: you can overwrite cases by simply hitting Run Case again with the same case highlighted, you don't need to make a new case if you don't want to save what was in the last one)

## 6. POST PROCESSING

General Things

When you run things in metric units, the post processor shows everything in Kelvin. You can change your units to Celsius in the thermal preferences window.

Transient Solutions

1. Moving through time. You can go through time step by step by clicking on the icons that looks like a contour bar and an arrow. It will go forward or backward depending on the arrow direction. You can also select a specific time by clicking the icon directly above them that looks like a pencil and a piece of paper. This will allow you to jump to a specific time step.
2. Animation. You can animate your results by clicking the Cycle Through Time icon a few down from the time step icons, its at the bottom of that toolbar and looks like a contour bar next to a circle.

## 7. MISC. FEATURES OF AUTOCAD THAT ARE USEFUL TO KNOW

This section contains information on features that will help with creating, visualizing, and analyzing your model.

Viewing and other Preferences:

1. *Thermal – Preferences*.
    a. Units Tab. When you first start AutoCAD, it will ask you what units you want to input. If you choose SI by accident, and you entered everything in English, you

can use this tab to switch to English. The "Don't Scale Model to New Length Units" if checked will do this. If it is not checked, the computer will assume you were inputting in SI units as you stated when you started, and it will convert these numbers to English (which is also nice if that is what you want to do).

b. Graphics Visibility. Here you can select what you wish to be visible on the screen and what you don't.
c. Graphics Size. Very useful, it allows you to change the size of objects displayed on the screen. You almost always want to do this for nodes because TD displays them pretty large by default, and it makes the display cluttered.
d. S/F Output. No clue.
e. Advanced. Also no clue.

Layers

Using Layers is like using groups in Patran. It will allow you to only display certain things, which is very useful.

1. General Layer Preferences and tasks
    a. Click on the format layer icon, or *Format – Layers*
    b. In the window that pops up, you can create or delete a layer, or make a layer current by clicking one of these buttons (you can also double click on a layer and that will make it current). The white window displays a list of all layers. The sun icon toggles visible/invisible, the next icon freezes/unfreezes a layer (freeze means that the layer cannot be selected or modified or regenerated, use this to speed up regenerations on complex models, otherwise it is not very useful), the next icon locks/unlocks a layer (will display the layer but you cannot edit it, although if a locked layer is current, you can add new objects to it), and the last icon selects a color for objects in a layer. Next to the lock icon are options for line style and line weight that may be used to further edit the display of objects in a layer.
    c. You select a few of the above options by using the layer drop down list on the upper tool bar next to the format layers icon. This is a quick way to set the current layer and turn layers on and off.
2. Adding Objects to layers and Changing an Objects Layer
    a. To add an object to a layer, first create a layer, and make it current.
    b. Any object you now draw will belong to that layer. If you want to put a new object in a new layer, create a new layer and make it current.
    c. You can change the layer a particular object is in by double clicking on it. This will bring up an object properties form, with one option being layer. Simply pick the layer you would like the object to belong to by choosing one from the dropdown list.
    d. I don't think you can assign an object to multiple layers.
3. TIPS for Layers
    a. Start creating layers from the beginning. It will make it easier in the long run especially for complex models. Remember, you can always change the layer a certain object is in.

b. When certain things are created, like making FE models from AutoCAD geometry (see Creating FE Meshes, #4) you cannot select things that the were created on or over. To avoid this, you may want to make a new layer, and set it as the current but turn off its visibility. By doing this, you can create whatever it is you want, but it will not be displayed.
c. You can change the current layer while in the middle of an operation. You cannot use the dropdown list however, and you must use the format layer icon next to it.

Modify Menu

The modify menu contains many useful tools for copying, translating, rotating and anything else you may want to do to objects, similar to the "Transform" action in Patran. It also has options for geometry manipulation like trimming, extending, fillet, etc., as well as change properties of objects.

Model Infrared or Solar Heat Lamps

Thermal Desktop can be used to model heating lamps consuming a known amount of power. This can be very useful to correlate experimental data to simulations. The following is the step by step method used for setting up lamps.

1. Draw your model first. Next draw the geometry for the heat lamps at the location you wish them to be. You can draw them to look just like the lamps you used, and put many nodes, or you can simply use a single node if you want to. Making this simplification is user discretion, but if you are using a long strip heater for example, you will want to model nodes along the length, and not just a single node at the center. You also want to specify the same node number for the nodes in the lamp. Make the ID's large, so that they are not the same as any other node in you model. You can do this under the Numbering tab in the thermal edit window.
2. Create multiple radiation groups. The purpose of doing this is so that the objects defined as the lamps will radiate to the other objects in the model, but they will not absorb any radiation from those objects. If you have multiple lamps that you would like to radiate from, but you do not want them to absorb radiation from any other lamp, you need to create as many radiation groups as there are lamps (there will also be the BASE group which is the default, and needed for the analysis, so you will have number of lamps + 1 groups really). To do this, select *Thermal – Radiation Analysis Groups…*, you may either copy the existing group (BASE) or create a new group. Copying will keep all geometry in the group, adding will create a group that has no current geometry in it. Say you have one lamp and some model that it is radiating to. Create a radiation group called "lamp" (you can copy if you wish) and hit OK. Select the model geometry, and hit the thermal edit icon. Under the radiation tab, you will now see two groups in the "Analysis Groups" section, the BASE group, and the LAMP group. For the model geometry, you want both of these two groups to be active, meaning that it is radiating top/out, bottom/in, or both. What this means is that the model will be able to "see" everything that is in the model, including itself, so it will absorb radiation from the lamps and radiation from itself (it will do this according to it's the direction you picked, top/out, etc). Next select the lamp geometry and click the thermal edit button. In this case, you want the lamp to see only itself, and not the model. You must set the BASE model to "Not in analysis group"

using the edit button. This means no radiation from the model will be absorbed into the lamp. Now it is clear why you may want to make a radiation group for each of the lamps in your model; if you put them all in the same group, all lamps will see each other and absorb radiation from each other. I do not think that this is possible without separate lamp groups, I tried doing multiple lamps with one group, and TD gives you an error message.

3. Now that your radiation groups are all set up, open up the case manager. Create a new case set or use the default, and click properties. Under the Radiation Tasks Tab, we will create a bunch of tasks. In the Analysis Group drop down window, select BASE. Click the ADD button. A new task will appear in the above window labeled as BASE. Double click on this task, or highlight it and click the Properties button. This task will determine how everything in the BASE group will be computed and outputted. Leave everything in the control tab and advanced control tab as is. In the Radk tab, enter the space temperature you would like, then hit ok. Now select LAMP from the Analysis Group dropdown box. Click the Add button, then double click on the LAMP group that appears above. In the control tab, change number of rays to 10,000 or 100,000 (can be anything, your discretion, it makes better calculations). Under the Advanced Control Tab, make sure the infrared check box is checked for modeling IR lamps, or solar is checked for modeling Solar lamps. Under the Radk tab, enter the same space temp as you did in the BASE group. Also check "Output as heating rates". Click the edit button next to this check box. Enter MAIN.#, where # is the node ID of the nodes in the lamp in the "from node text box". Enter the power consumption of the lamp. Choose absolute (this will take the total wattage you give and distribute it across the area of the lamp surface) or flux (watts/area) and then hit ok. Hit ok again. Repeat this process for each lamp group, each time entering the specific lamps node numbers into the Output as heating rates dialog box.
4. You are now ready to run the analysis. Set the remaining conduction properties in the S/F Tab as you like and run the case (it is a good idea to reduce the default values in the Convergence Criteria section of this tab for complicated solid models; except for Max iterations which you want to increase to maybe >50,000). For info on how to do this see the Analysis section.

***General tips for Thermal Desktop***

- Was very helpful in saving time to have a default TD file set up, with material properties and layers already defined, before starting
- Slows process to have to wait for both Mechanical and Structural
- Option of adding MLI in TD was a great time-saving modeling option
- Manual twisting of radiator to define optimal position is tedious but not too time-consuming (MDOB may be able to help)
- The thermal desktop nodes must be re-sequenced manually after any operation that changes or adds node numbers.
- To resize the results display: post the viewport and click the button to "Regen shade". Then click the MODEL button at the bottom to PAPER., and you can move items in the display (spectrum, etc).

- There is no way to display in m2 from a millimeter model.
- To rotate after a re-scale: In 3D rotate mode, right click, click “more”, and click “zoom extents”. (so that center and rotate scale is set correctly, and model doesn’t careen off the screen).
- To view the model from the sun: go to model checks and “view model from”.
- To see orbit info, select the orbit in the orbit display and type ‘list’.
- Run in a clean directory.
- To get rid of the annoying SDIMODE behavior where it won’t close drawings, go to Tools…Options…System, uncheck “Single drawing compatibility mode” and change to “Show traditional startup dialog”

## *Short Guide to MSC.Patran Thermal*

1. You will need the PATRAN location (e.g., /mirror/msc/patranXX/bin) in your path. You will also need access to the license or license server.
2. On UNIX, type "p3 &" (without quotes -- the & puts it in the background). On NT, just start by double-clicking on a db file, or on the PATRAN executable.
3. Start a new database with a template that includes PATRAN THERMAL.
4. If necessary, set the P/Thermal analysis preference. If you are picking up a structural model that was built from a NASTRAN only template, type "load_pthermal()" in the PATRAN window command line to load and set the P/Thermal analysis preference.
5. Determine what set of units you will use -- must be internally consistent, such as SI (W, m, kg, K, sec), or English (Btu/hr, ft, lb, F, hr). Units in all the following places must be consistent:
    - P3 model geometry length units -- as converted using the conversion factor in analysis menu
    - P3 model material properties -- either as manually input, or as read from materials file defined in analysis menu. It will ask you for this file again if you run P/Thermal by using patq option 4.
    - P3 model calculation temperature scale
    - P3 model boundary conditions
    - TRASYS files, as converted (if used)
6. Determine your group naming convention (e.g., GEO_sidecap, FEM_sidecap, are what is often used in our branch to keep FEM and geometry separate).
7. Create or import geometry
    - If you import from Pro/E, see “General Tips for Importing and Meshing Pro/E Geometry in PATRAN”
    - If you import from a Pro/E assembly model, the parts will be placed in groups by their part names (if you have the appropriate toggle switch on in Pro/E Options).
    - If you import a Pro/ECAD model, see “Thermal analysis of electronics board in PATRAN”
8. Define materials -- Note, you can see the materials' names by using the PATRAN shareware thermal tool for viewing the materials library, but you cannot see the actual properties from there. To see properties you have to view the text file (mpid.*) in

/patran/p3thermal_files/lib/. It's a good idea to check that the material property you are going to use extends over the temperature range of interest.

9. Assign properties using your desired materials. Note: it is usually a good idea to assign properties and boundary conditions to the geometry, so that you can re-mesh without needing to change anything else.
10. Add boundary conditions for heating and temperatures, etc.
    - See section 4 of the PATRAN on-line help for specific examples.
    - Most often the wisest choice will be to assign BCs on your geometry, so a re-mesh won't cause as much work.
    - For time- or temperature-dependent BCs, you can either use a field, or a time-table within the BC, or define the MICRO function in an external text micro.dat.apnd file. A warning for fields -- if you use the built-in micro-functions to create your field (some of which are very useful), be VERY cautious about using the Hermite polynominal interpolation -- it can do some radical mis-interpretations of your data if you are not careful.
    - To add convection or radiation to 1D bars, see http://www.mechsolutions.com/support/knowbase/PATRAN/tan/tan3251.html (external link)
11. Mesh and equivalence.
    - It is often a good idea to do all the material associations and BC applications before you mesh, because you may find places that you want to break your solids down to apply BC's, and if you have already meshed then you will have do delete and re-do the mesh.
    - You can only use the IsoMesher (auto hex mesher) on "blue" (native) solids. If it's a B-rep solid (imported from Pro/E) you need to use the TetMesher.
    - Don't equivalence two parts together if you have a contact resistance between them.
    - It's a good ideas to make separate groups for each meshed part (e.g. FEM_part_name). Then you can have one group called All_FEM for displaying results etc.
    - If a solid won't mesh, try surface meshing the outside first (then you can delete the surface mesh later).
    - If a surface won't mesh how you want, try mesh seeding the edges.
    - For PATRAN Thermal, you can only use the simplest element types (i.e., Tet4, Hex8, Wedge6).
    - If you want a mesh to go across non-congruent geometry (such as two plates with non-matching edges), you can associate (for example) the shorter edge of one plate with the side of the other that it is touching. Then the mesh will run nicely across the interface.
12. Check your BC's and materials on your mesh. This is important to make sure that your BCs are applied where you want them, and that your FEM is truly associated to the base geomety, especially if you have done anything fancy like re-associating FEM to solids.
13. If you need to add user logic, see the on-line help manual and create your own ulib file from edited subroutines.
14. If necessary, make any modifications to the Default load case.

    - Currently, load cases are not fully available as an option for thermal analysis. Only the Default load case is used. You can, however, create a hot or cold case by selecting the BCs which go into the Default load case. For example, you can have BCs for MinQ & MaxQ. The user goes to load cases & deselects the MinQ BCs for the Max Q case & visa-versa.
    - Something to note is that the PATRAN default is to add all BCs to the Default load case, and to combine them using 'Add'. This means that any BCs that touch at a boundary will be added together at that boundary. You can change this by altering the BCs in the Load Case to combining in a priority order.
15. Run analysis using analysis form -- must have FORTRAN on the machine you are using. This will create nr*.nrf.* result files.
    - It is best when you are going to run an analysis if you started PATRAN in the directory of the db you are going to run from. If you started PATRAN in a different directory, then closed that db and opened the one you want to run from, it will get confused about directory locations.
    - If you have a micro.dat.apnd file you want it to pick up in the run, you must have at least one field in your db (can be a dummy). It will pick up mat.dat.apnd and template.dat.apnd automatically. It will also pick up a convec.dat.apnd if you have one.
    - In general, it is best to run in an empty directory so that files from older runs do not get used. For example, if you have only one temperature BC in an early run, and then delete that BC, the temperature BC file will still exist in the analysis sub-directory, and WILL get used if you run again into that same sub-directory, unless you go back manually and delete the files. Easiest is to always run in a new directory job name.
    - If you ran a transient, you can check the times for each output step, even though they may be binary files, by doing a 'more' on the file and looking at the time definition at the top, which is in (messy) text format.
    - If you are going to have a lot of runs, you can sort your result files into directories with helpful names, and still be able to view them.
16. Read results and display using analysis/results forms. Use PATRAN Shareware utility for pulling in multiple files.
17. If desired, create time-temperature plots using " Additional PATQ Utility Options " in patq from command line.
18. If desired, edit qin.dat, mat.dat, template.dat, convec.dat, gap_convec.dat, tfix.dat, temp.dat, qbase.dat, etc. to alter variables, and then re-run using patq (from command line), options 4 and 5.

Notes on operation:

- *Analysis can be run from within PATRAN (graphical window), or in command line form by typing 'patq' in a UNIX or DOS window*
- ***NT:***
    - *Run patq interactively by simply typing "patq" in a DOS shell*
    - *Patq can be run in batch mode (although you can't close the DOS shell without losing the run, you get the prompt back)*

        - *Type "patq < patq.ses" where patq.ses (name optional) contains the commands you would give patq manually*
        - *you can use a bash (UNIX emulator) shell if all you want to do is use auxillary options in patq, but if you want to actually run the solver you must use a DOS shell*
        - *you can set up buttons on your toolbar to start DOS shells where you normally need to run jobs*
        - *You can stack multiple jobs in a command file and then type (for example)*

          **at 4pm "C:\AT_patq.cmd"**
          *where the time and command name and location are user options*
- ***NT notes for running with ulib FORTRAN:***
    - *combine all u*.f files into ulib.f (can be done in bash shell using 'cat u*.f > ulib.f')*
    - *compile ulib.f in DOS shell using 'ulib ulib.f' (makes ulib.lib)*
    - *place ulib.lib in in analysis sub-directory*
    - *run patq from within PATRAN into identical job sub-directory name or from DOS shell*
- ***UNIX:***
    - *To run patq in batch mode (so that that you can log out):*
        - *Type "patq & patq.log &" where:*
            - *patq.ses (name optional) contains the commands you would give patq manually*
            - *patq.log (name optional) is a log file that will contain all the run information*
            - *the final "&" puts it in the background*
        - *If you want to run more than one job, you have to guess how long the first one is going to take, and then use a command like:*
          **(sleep 6000; patq & patq.log ) &**
          *for the next run (where 6000 is the number of seconds you want before it starts)*

***Including TRASYS orbital fluxes in Patran Thermal***

1. Do everything above up through meshing and checking.
2. Create a template.dat.apnd file to define your TRASYS (and P/thermal, if desired) optical surface properties.
3. Apply the optical property templates to your surfaces. Make sure all the normals are pointing the way you want the active surfaces.
4. Translate from P/thermal to TRASYS model. This can be done either from within PATRAN by choosing 'Output TRASYS model' on the analysis form, or (perhaps easier) from patq on the command line, once you have made a neutral file, by selecting to create a TRASYS model, in 'Additional PATQ Utility Options.' In order to have the back translation work correctly, all the files generated by the VIEWFACTOR run must be saved (see P/thermal on-line help).

5. Decide if you are going to use P/Viewfactor or TRASYS for the radiation conductors. If P/Viewfactor, you will have to run the analysis for that before executing P/Thermal for thermal solution. If TRASYS, you will need to import your TRASYS Radk's, and worry about units for radiation conductors and the space node number in PATRAN (see notes below, also in home analysis page).
    - If you use P/Viewfactor to calculate Radk's, it may generate some huge files when it runs, depending on the size of the model. If the model has been imported from Pro/E, you may not be able to get around this by defining symmetry planes and using similar tricks to simplify the radiative calculations.
    - When running the TRASYS orbital flux subroutine QOCAL and calling QODATA, the variable QOTYPE must not be 'BOSI' (giving solar and IR broken out separately), but must be 'BOTH'.
    - The translator currently translates any surface as a TRASYS 'POLY' surface type. However, if the surfaces are actually regular QUADs, search-and-replace editing can be used to change the TRASYS model file to QUADs, which will cut solution time appreciably. The results can then still be brought back into P/Thermal for solution, or can be incorporated in the translated SINDA model.
        - Any editor can be used to do the search and replace to change 'POLY' to 'QUAD'; if vi is used, it can be done with
          :%s/POLY/QUAD/g
        - Currently, vi can be used to remove the P4 line (which needs to be done since the QUAD definition only uses three points)-- to do this (i.e., match lines beginning with 6 spaces then "P4=" and replacing to the end of the line, no matter what the rest of the line is):
          :%s/^ P4.*$/C/
          (where . is a single character match, * extends it to any number of characters, and $ defines that the match goes to the end of the line).
          If you want to take out paragraph returns:
          cat filename | egrep -v '^C' > filename.out
          (where this would take out all lines beginning with C from filename, and leave the results in filename.out).
        - The nedit editor can also be used.
    - It's a good idea to have TRASYS output results in the unit set you would like to use in P/Thermal (using the factors in the RKDATA and QODATA lines).
    - The P3Thermal TRASYS reverse translator treats TRASYS ambient (space) node radk's differently than other surface-to-surface radk's. In the forward translation to TRASYS from P3THERMAL each element number from P3THERMAL is multiplied by 10 as it is sent out to the TRASYS input deck as a surface. If the enclosure references an ambient node then that node is added in a comment card at the end of the TRASYS input deck. When you create your operations block in TRASYS you have three options with regard to numbering the SPACE node:

        The preferred option is to use "0" as the default SPACE node number; this will force TRASYS to use 32767 as the space node number. On reverse translation from TRASYS to P3THERMAL there is logic in the converter to detect incoming 32767 references and to convert those

references to the original P3THERMAL ambient node number which was included in the comment card.

- The second option is to hard code "32767" as the TRASYS SPACE node number. The consequences are the same as option 1.
- The third option for numbering the SPACE node is to number it the same as your ambient node ID in P3THERMAL multiplied by 10. This will circumvent the 32767 logic and allow the reverse translator to strip the trailing "0" without mangling your original P3THERMAL space node number.

2. Run TRASYS to generate orbital fluxes and radiation conductors (will get filename.bcd).
3. Run translation back from TRASYS output to P/Thermal input using patq 'Additional PATQ Utility Options'.
    - The translator will take all values from your one file (filename.bcd). If you are not working in Btu's, feet and hours, and you did not choose to have TRASYS output results in the units you are using in P/Thermal, then you will need to give the translator the factors you would like to use. The translator divides TRASYS heats by the factor you give it, regardless of how the wording sounds. The time multiplier you give it is used as a multiplier.
    - You will get tramic.dat and traqma.dat for transient orbital heating, trqbas.dat for orbit average heating, and trarst.dat for radiation conductors (if you chose to translate them).
    - It's a good idea to check these files to make sure they ended up in the units you wanted. Tramic.dat will have time vs. heat arrays, and trqbas.dat will have psuedo-static heating averages. To check the files, you will have to find the TRASYS surface(s) that relate to a particular PATRAN node. To do this, check the TRASYS input file for the surface numbers relating to a particular PATRAN node. Divide the heating on those surfaces to their respective nodes, and sum all heating to the node you want to check. Then check that value from the P/Thermal input files against what it was from the TRASYS bcd file. In other words, check that the heat on node X in the tramic.dat and trqbas.dat files is the same as it should get from the TRASYS surfaces it is associated with.
4. Run P/Thermal just far enough to create a qin.dat file. To do this, in the PATRAN analysis menu, in 'Submit Options', select everything necessary in the first section, but do not select 'Execute Thermal Analysis' in the last section.
    - One nicety is to create two qin.dat files, one for transient runs (say, qin_transient.dat), perhaps with a restart file name defined, and one for steady-state runs (say, qin_static.dat). You can do this by hitting Apply in the analysis form twice, each time with the parameters you want. However, you must re-name the first qin.dat you create before you hit Apply the second time, or the first qin.dat will be overwritten. If you don't know the restart file name, you can just put in a dummy name, as you will be able to edit it later. Make sure for the transient run you define the start and stop times and print interval as desired.
5. Edit the qin_static.dat so that it ignores traqma.dat and tramic.dat (comment out those lines).
6. If desired, run the steady-state case to create a restart file you can use for the transient run. (Normally, one would want to start a transient case from a steady-state result case in

order to get a realistic transient). To run, copy qin_static.dat to qin.dat. Run patq, option 4 then option 5. You must do this in the directory where qin.dat, your trqbas.dat file, and all analysis files are located.

7. Edit the qin_transient.dat to change the restart file name to be the nrf file that was created in the previous step (if necessary), and comment out the trqbas.dat line, so average heating will not interfere with the transient heating array definitions.
8. Copy qin_transient.dat to qin.dat. Run the transient case (again, must be in the directory where files are located). Run patq, option 4 then option 5.
9. Read results and display using analysis/results forms.
10. If desired, create time-temperature plots using " Additional PATQ Utility Options " in patq from command line
11. If desired, edit qin.dat, mat.dat, template.dat, convec.dat, gap_convec.dat, tfix.dat, temp.dat or qbase.dat to alter variables, and then re-run using patq (from command line), options 4 and 5.

Alternate method:

- Create a TRASYS file and run.
- Use Excel macros to put TRASYS .bcd output file in micro.dat.apnd format.
- Apply boundary conditions to PATRAN model using MICRO number.

# Thermal Interface with Mechanical/CAD Station

Parts will normally be pulled the CAD tool Pro/Engineer. A couple of simple actions in Pro/E are valuable for the thermal engineer to know.

- To open a part (prt) or assembly (asm) file, double-click on it (assuming you have Pro/E installed).
- To rotate, use middle mouse button.
- To see as solid shaded, use solid shaded cube icon in menu bar.
- If the part has more detail than you want to import into your thermal model, use the suppress feature described below.
- To export as STEP, use File…Save a Copy..and select STEP (.stp) format.
- If you suppress features and then save as Pro/E or STEP format, the part you import will not have the feature.

***To suppress features***

- Select a feature in the Model Tree or in the graphics window and right-click. A shortcut menu appears.
- Click Suppress. The selected feature and its children are highlighted in the Model Tree and a Suppress message window appears. Note: You can also use Edit > Suppress after selecting the feature.
- Click OK to suppress the feature and all the children of the feature or click Options (if available) and the Children Handling dialog box opens. Note: Once the feature/layer is suppressed it will not appear in the Model Tree.

***To resume features***

- Click Edit > Resume.
- Click on one of the following commands:
    - Last—Resumes the last feature you suppressed
    - All—Resumes all the features you have suppressed.
    - Select—Resumes the selected feature.

## *Thermal Interface to CAD Using Thermal Desktop*

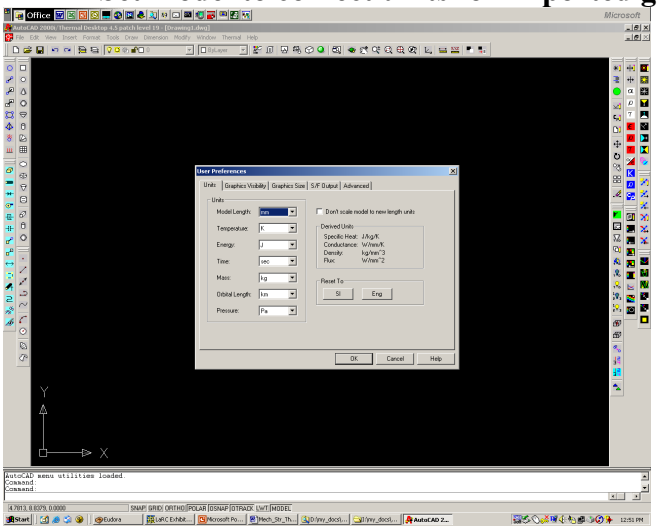
*This describes the interface between the Mechanical CAD station running Pro/Engineer, the Thermal station running Thermal Desktop/RADCAD, and the Structural station running Patran and NASTRAN.*

Summary – you can pull CAD geometry in via ACIS, IGES and STEP – STEP is the most robust. You can pull a mesh in from NASTRAN format, which can be created in Patran or Pro/Mechanica. General geometry import from Pro/E gives you only surfaces that you cannot immediately mesh – you have to snap native Thermal Desktop surfaces on to them to mesh. If you want a fine mesh, the methods described under the sections "Importing from NASTRAN" or "Importing Pro/Mechanica (NASTRAN) mesh of a Pro/E part" may be preferred.

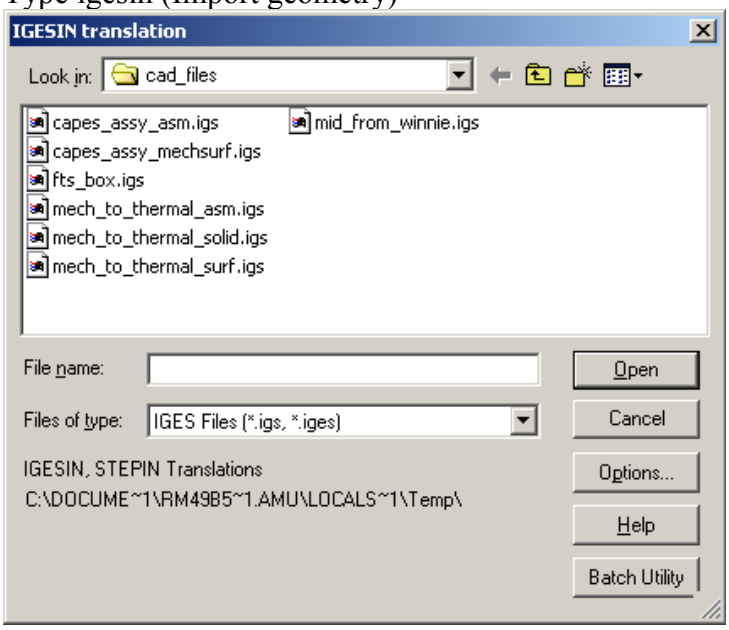
***Pulling in IGES***

*This pulls in boxes and flat surfaces nicely but doesn't handle surface orientation. That has to be changed manually.*

- **Set model to correct units for imported geometry (e.g. mm): Thermal, Preferences.**



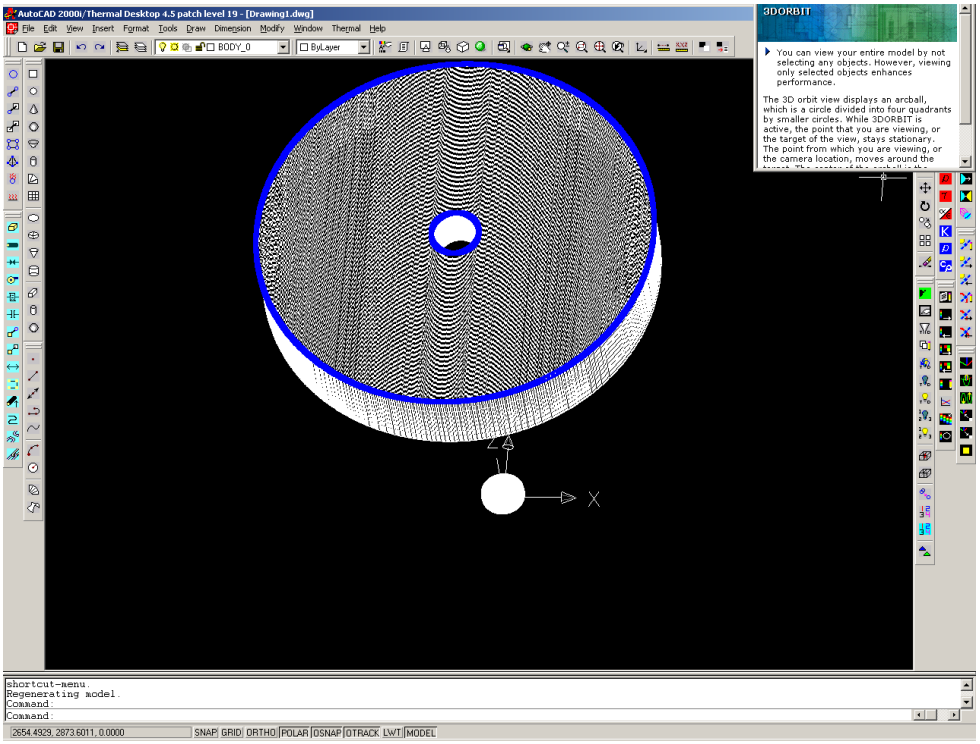
Type igesin (Import geometry)




Set options as desired (for surfaces, default options work fine)

Select desired file

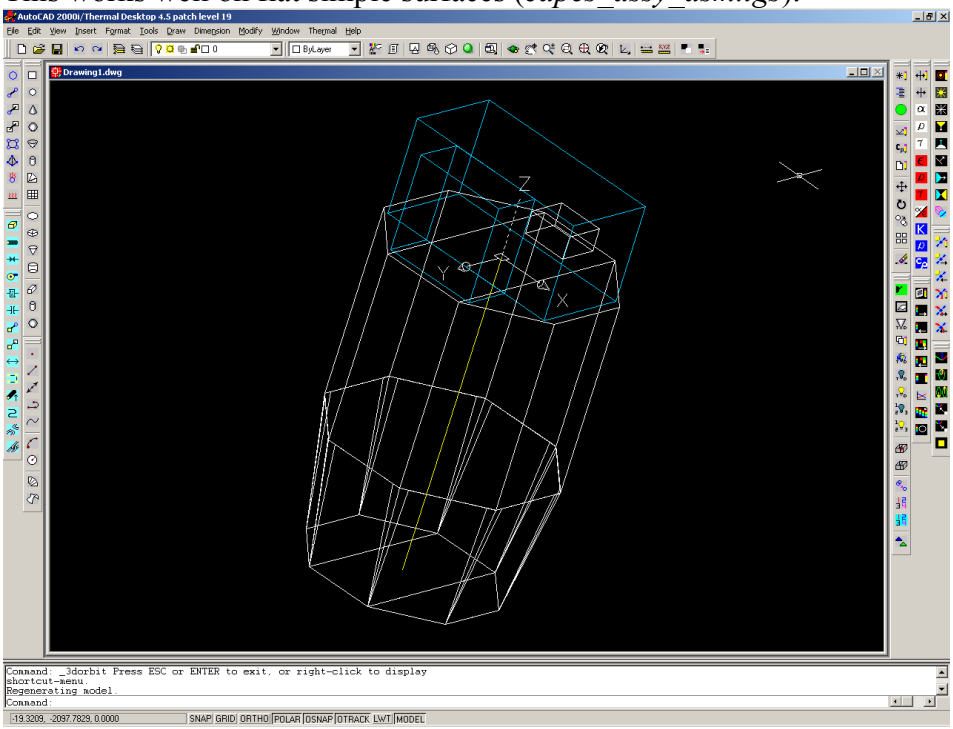
This is too detailed on a curved surface….




Set model back to desired units (e.g. m) (uncheck “Do not rescale model to new units”)

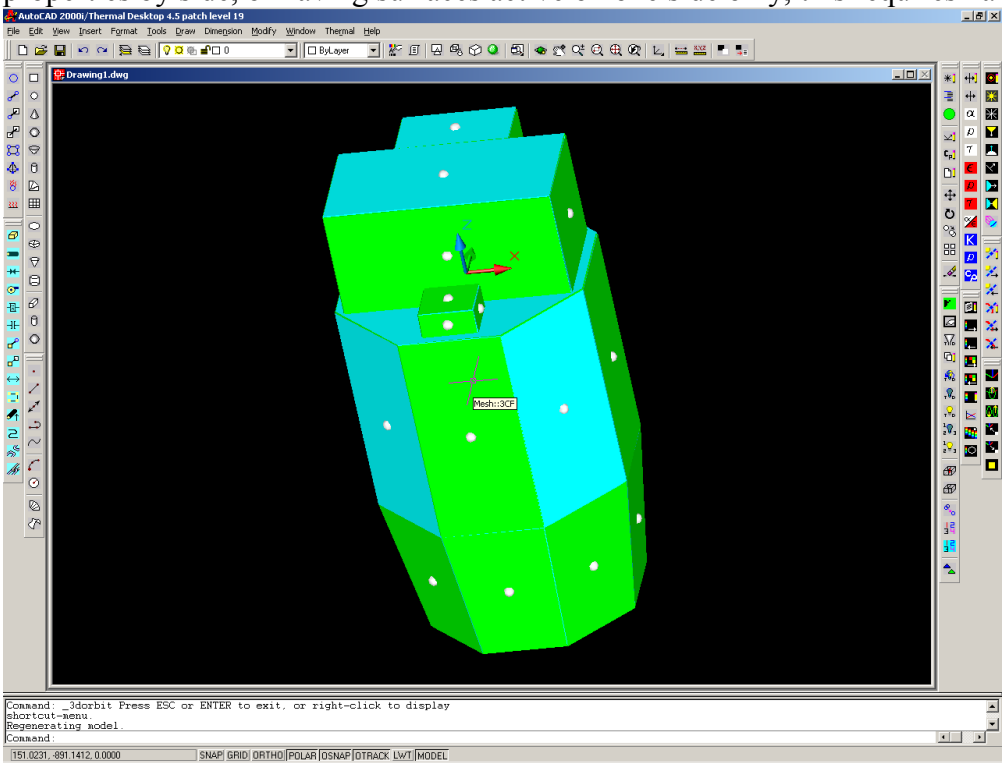
IGES solid pulls in nothing – use IGES surface files instead.

This works well on flat simple surfaces (*capes_assy_asm.igs*):

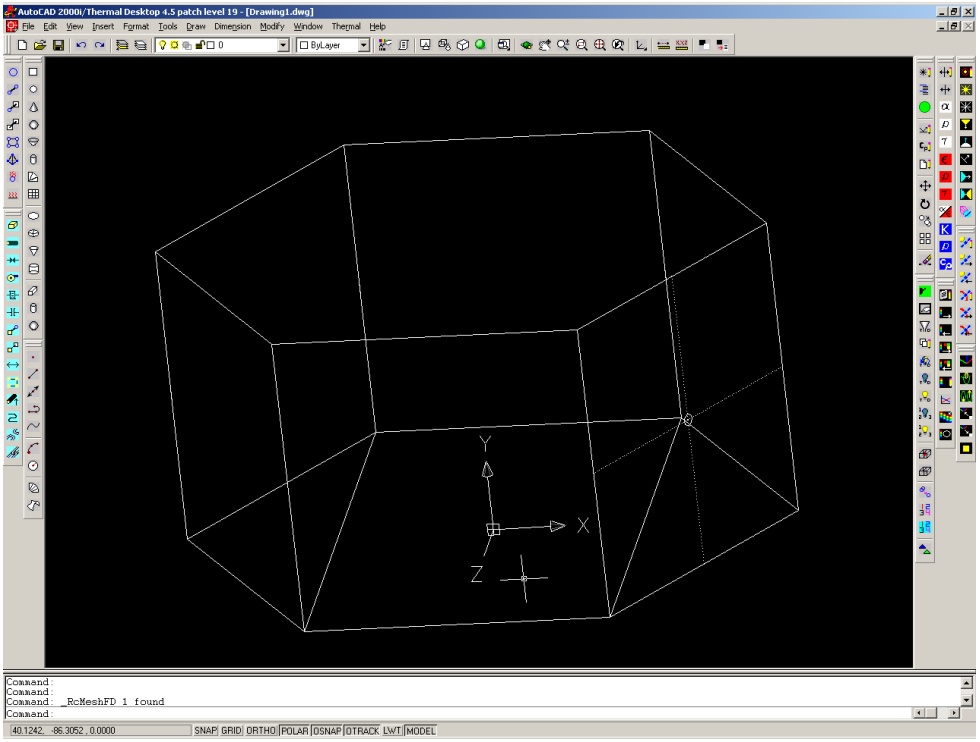



After bringing in the geometry, you have to go through the below process under "To make Thermal Desktop surfaces" to make them into Thermal Desktop surfaces.

However, the surface normals will not necessarily all point the same way, so if you are setting properties by side, or having surfaces active on one side only, this requires hand-work to change:



Another method is to bring in mid-plane IGES surfaces as created by the structural analyst in PATRAN:




After bringing in the geometry, you have to go through the below process under "To break surfaces up**"** and "To make Thermal Desktop surfaces**"** to make them into Thermal Desktop surfaces.

## ***Pulling in ACIS***

Set model to correct units for imported geometry (e.g. mm)
Type acisin or go to Insert…ACIS File
Set model back to desired units (e.g. m) (uncheck "Do not rescale model to new units")

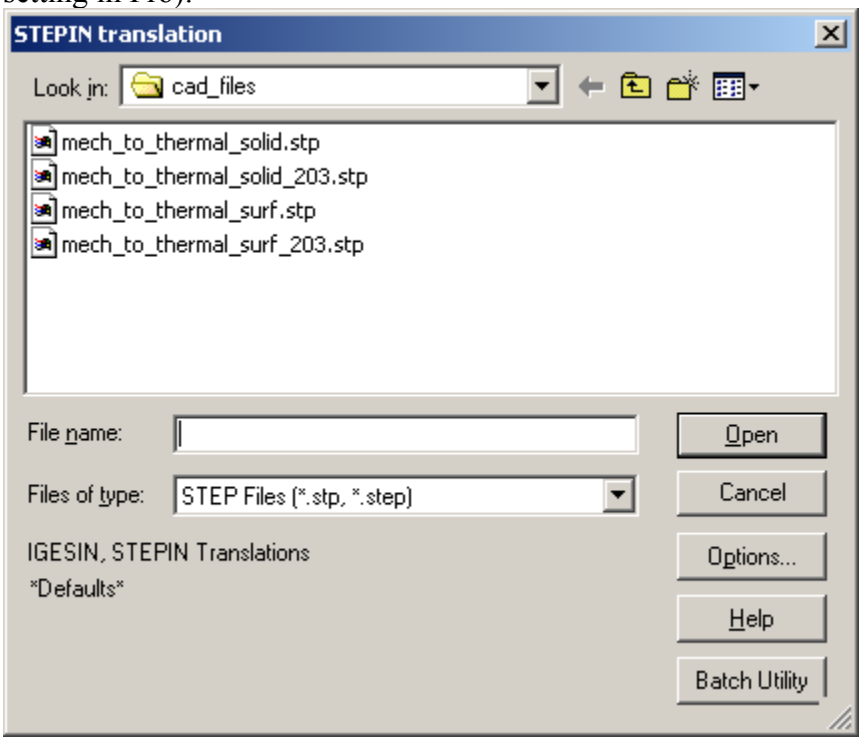
This pulls in solids and surfaces from a Pro/E file. You can save both part and assembly files in ACIS format. (In Pro/E Wildfire Do Save a Copy...and pick ACIS (*.sat))

After bringing in the geometry, you have to go through the below process under "**To break surfaces up"** and "**To make Thermal Desktop surfaces"** to make them into Thermal Desktop surfaces. Alternatively, you can snap TD surfaces and solids to the imported shapes.

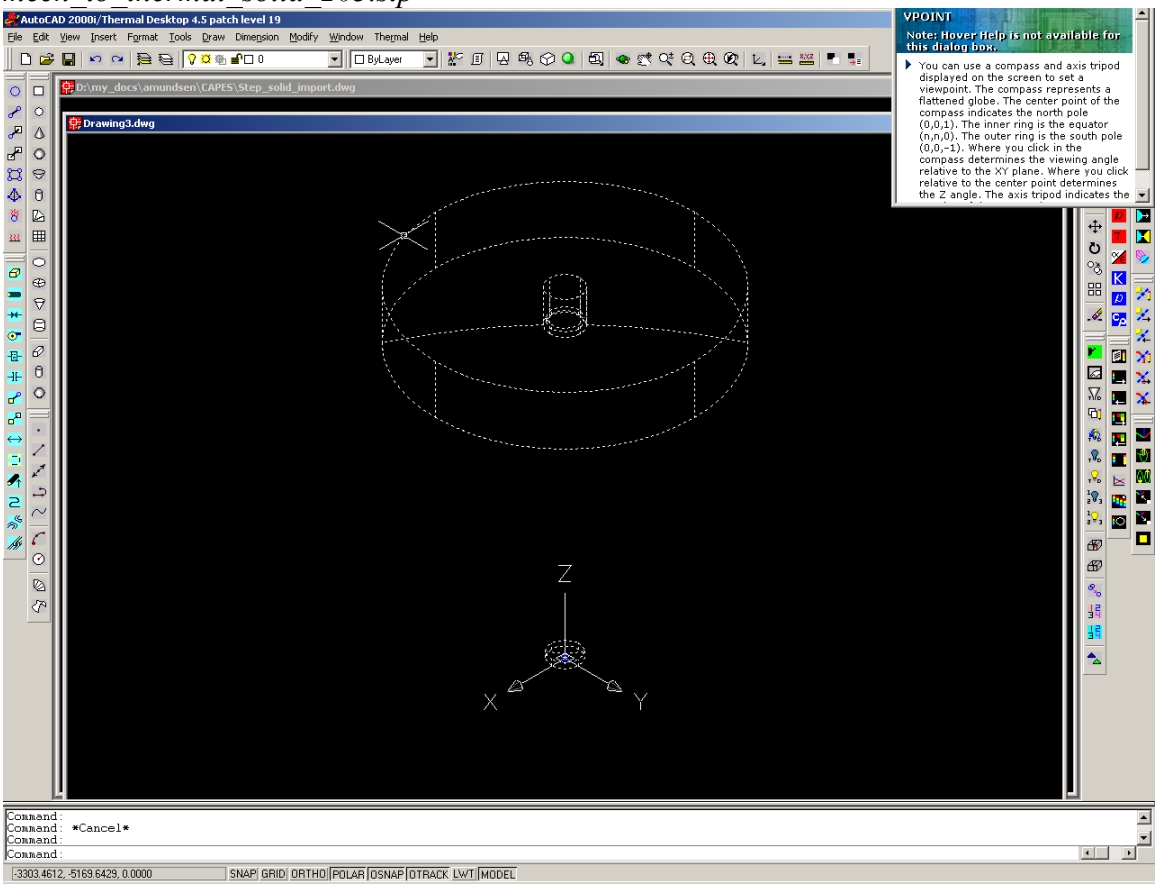
## ***Pulling in STEP***

*This pulls in curved surfaces but makes them all triangle surfaces. Triangles cannot be meshed more finely. Active surfaces appear to be all in same direction on the outside of a solid.*
Set model to correct units for imported geometry (e.g. mm)

Type stepin (Import geo) – pull in STEP 203 file from Pro (no difference between 214 to 203 setting in Pro):

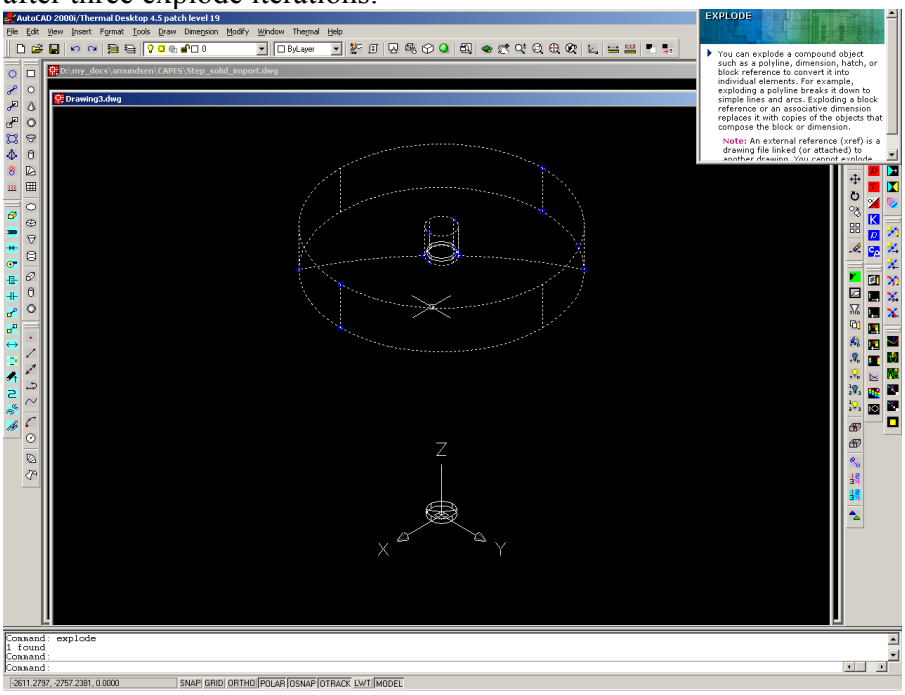



*mech_to_thermal_solid_203.stp*



Set model back to desired units (e.g. m) (uncheck “Do not rescale model to new units”)

After bringing in the geometry, you have to go through the below process under “**To break surfaces up”** and “**To make Thermal Desktop surfaces”** to make them into Thermal Desktop surfaces.
after three explode iterations:

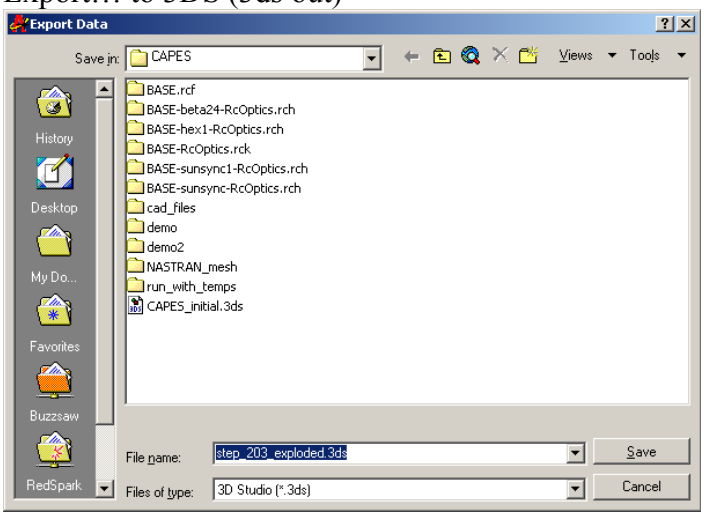


***To break surfaces up***

Select everything
Modify… Explode (until you have surfaces)
If original model is not in desired units, easiest way will usually be to change unit sets at this point:

Set model back to desired units (e.g. m): Thermal…Preferences, set length to "m" (uncheck "Do not rescale model to new units")

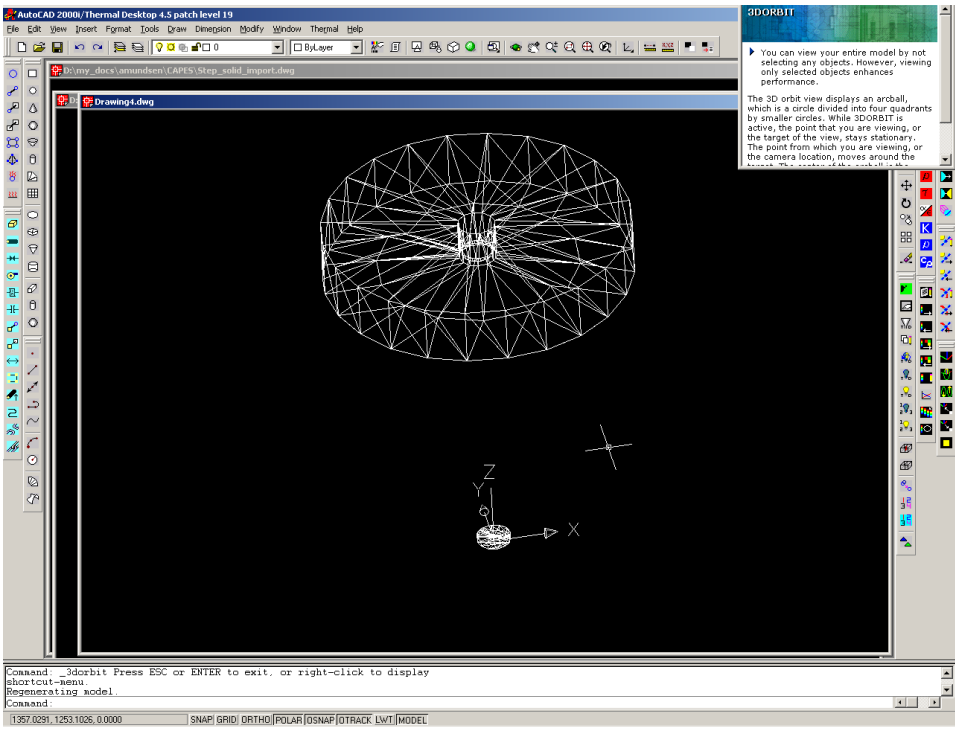
Export… to 3DS (3ds out)




Select all (or type "all" at prompt)
ACI, chg Auto-smoothing if necessary

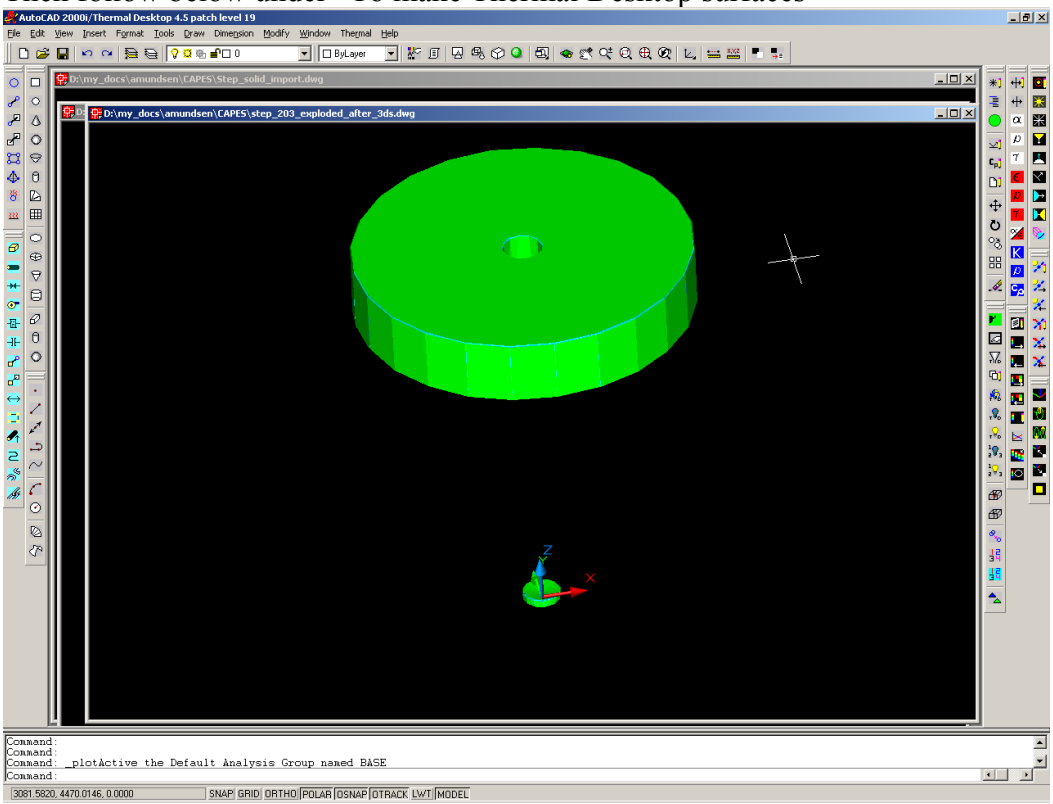
Close drawing
Open new drawing
(Change unit preferences if necessary – or change unit preferences before making the 3ds file)
Insert…3DStudio file

Add All




**Modify…Explode (all) before making TD surface**

Then follow below under "To make Thermal Desktop surfaces"



***To make Thermal Desktop surfaces***

Thermal…Surfaces/Solids…From AutoCAD surface

Select all

For single-sided, check "Use same Ids both sides", then have active Top/Out

Give it thickness, material, optical material

Re-sequence Node Ids – make sure you have all elements posted so that all actually get selected

***General notes on working with imported geometry***

Convert surfaces/boxes in groups:

- Grab everything you want to be 2-sided surfaces with the same thickness, and convert them together to TD surfaces.
- Grab everything you want to be 1-sided surfaces with the same thickness, and convert them together to TD surfaces.

If you need to grab several surfaces and change a variable on them that is not already the same (e.g., you want to change them all to be single nodes, i.e. use the same node number for both sides), then grab them all, do Thermal…Edit, and uncheck the box, then grab them all again and then check the box.

***Importing from NASTRAN***

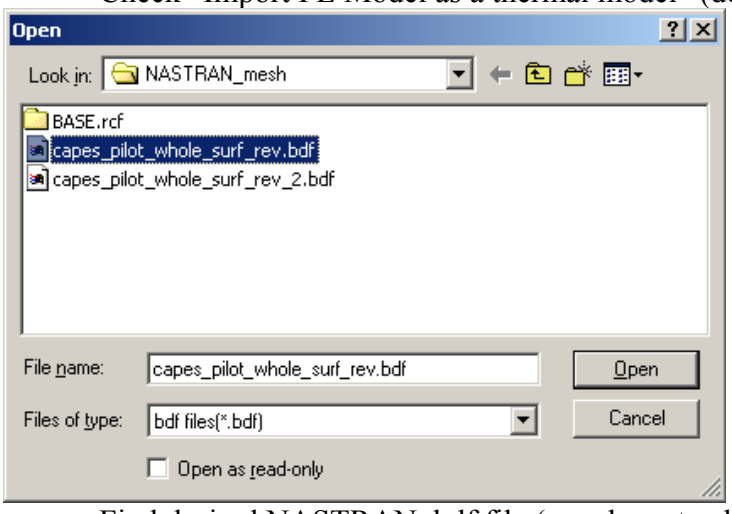
A preferred method in many cases will be to take the meshed geometry from the structural station. This avoids all the manual steps of re-orienting surface normals, exploding grouped geometry, creating Thermal Desktop surfaces, and meshing.

*Importing from NASTRAN: Steps*

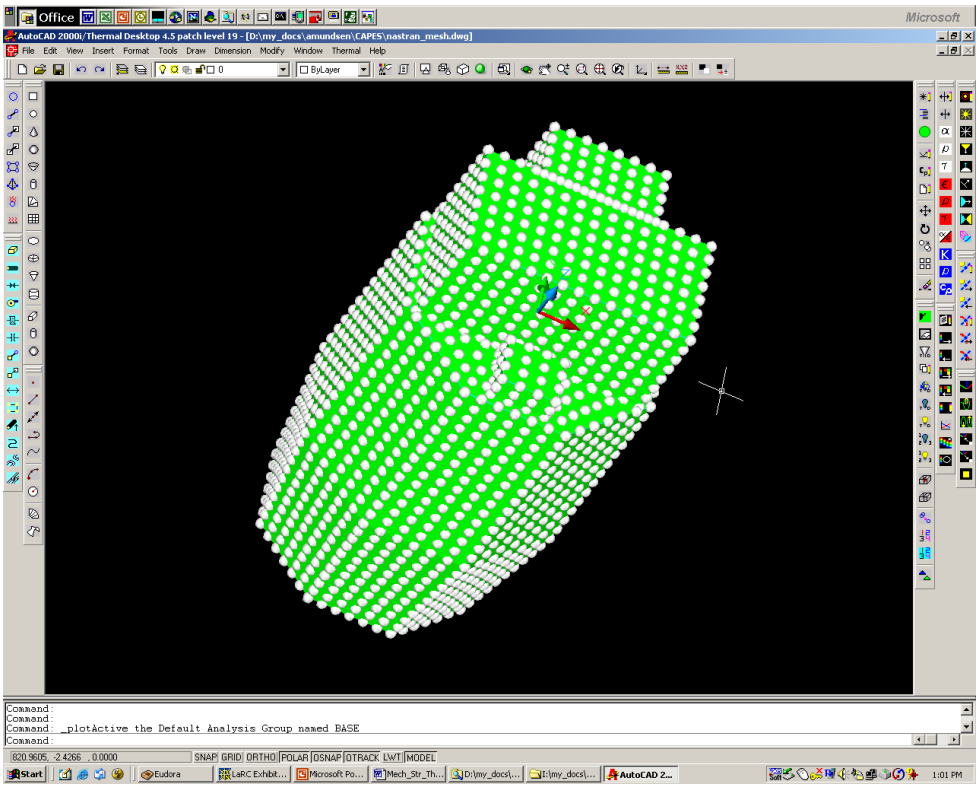
Set model to correct units for imported geometry (e.g. mm)

Thermal…Import…NASTRAN

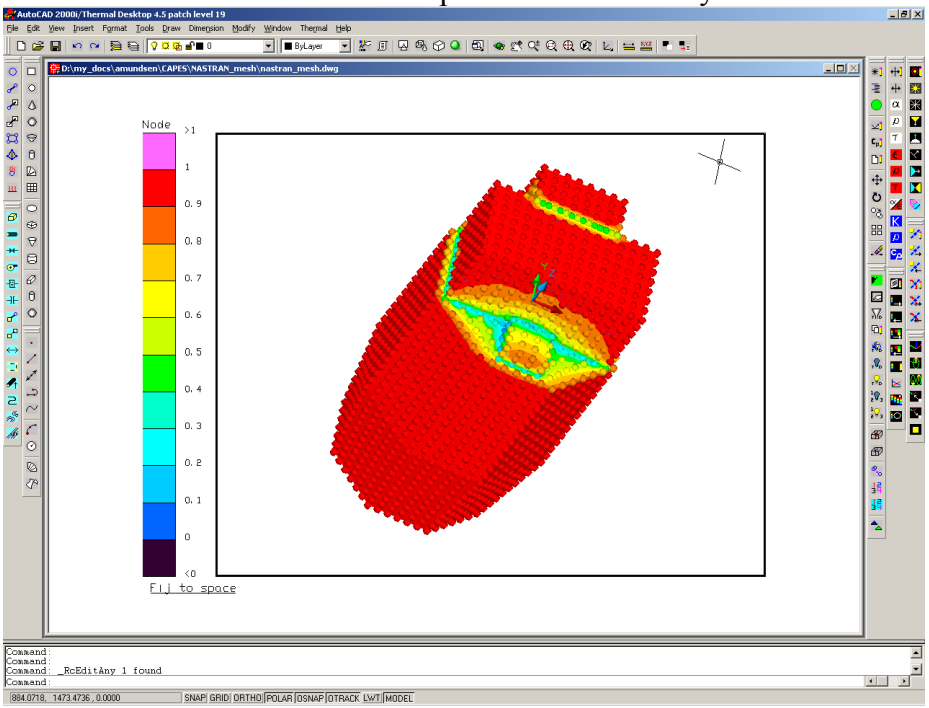
Check "Import FE Model as a thermal model" (default)




Find desired NASTRAN .bdf file (may have to chg file extension)




Set model back to desired units (uncheck “Do not rescale model to new units”)
If surface normals are oriented correctly in NASTRAN, and only one side is desired in Thermal Desktop, this leaves all the active surfaces in the same direction. The model, however, can now not be remeshed by the thermal analyst (which is true anyway for most imported geometry).
Thicknesses of surfaces come in correctly.
Calculations can be run on this imported model directly:



It will be the easiest to work with the components if they are brought in separate files from NASTRAN (Structural person can save as groups from PATRAN). To do this, make a new layer for each part and make it current before the file is brought in (remember new items will only go on the layer that is current). Then that entire part mesh will be on a single layer, which will make display and selection by part much easier.

***Importing Pro/Mechanica (NASTRAN) mesh of a Pro/E part***

Open Pro/E Wildfire
Open part (or assy) file
Do Applications…Mechanica (you must have Mechanica installed)
Check on "FEM Mode" in the pop-up dialog box
Go to Thermal (can also do in Structural)
Select Mesh, Create, Solid (or Boundary), Start
    It will create the mesh (check it or change it if you want to) -- it does only tet for solids
    Hit close on the Check window
Do Analysis, FEM Solution, under SOLVER pick MSC/NASTRAN
    Analysis: leave it on Structural
    Element Shape: Linear (or can use parabolic)
Leave it on Output to File
    Hit OK – it will export the NASTRAN mesh file as a *.nas file
Close Pro/E

[Note: In the past, you have had to edit the NASTRAN file to have each of the following lines not carry over to a second line, but this is currently not necessary, e.g.:

```
$ Global Coordinate System of the model
CORD2R,1,0,0.,0.,0.,0.,0.,1.,
,1.,0.,0.  ]
```

Open a Thermal Desktop file
Go to Thermal…Import…NASTRAN (make sure you set units first)
    Select file by changing to *.nas files – pick your desired file
You have a meshed model!

Boundary conditions must be applied by using one of the filters to select element faces as necessary, since you don't have the base geometry.

Right now display of a complex model is pretty slow, but I think it will be better in Mechanical Desktop or AutoCAD 2005.

***Importing Nastran Bulk Data into Thermal Desktop: Lessons Learned***

This section is courtesy of deborah.m.lemons@nasa.gov at NASA JSC.
Integrated structural/thermal analysis is facilitated by model sharing between disciplines, particularly FEMs. Thermal Desktop is one of several thermal analysis programs that are capable of importing finite element networks and converting them into thermal networks. (Other such programs include SINDA/ATM, FEMAP TMG, and IDEAS TMG.)

MSC.Nastran is a widely used stress analyzer whose FEMs, compiled in "bulk data" format, are recognized by the Thermal Desktop translator. (Other recognized, but less-widely used, formats include FEMAP neutral, IDEAS universal, and STEP-209.) Following a successful thermal analysis in Thermal Desktop, temperature results for FEMs that are also used for mechanical stress analysis can be fed back into the Nastran solver, thus closing the loop on an integrated structural/thermal stress analysis.

During translation, network grid locations and many Nastran element types are retained. However, due to inherent differences between thermal and structural analysis methods, discontinuities are likely to occur when translating an FEM that was specifically created for modeling mechanical stress. (In other words, certain modeling practices suited to mechanical analysis do not appropriately treat thermal phenomena). Furthermore, because model translators must struggle to support ever-changing technology, they suffer from a perpetual need to become increasingly robust, seamless, and user-friendly. This report, based on user experience, contains helpful information for addressing these kinds of difficulties encountered upon importing Nastran bulk data into Thermal Desktop. The intent is to promote correct and efficient translation and, ultimately, a better final product.

*Getting Started*

Analysts can increase the odds of a successful translation by becoming familiar with common Nastran element types and bulk data entry formatting (see Reference 1). (Examples of Nastran element types that are relevant to thermal modelers include CTRIA3, CQUAD, CTETRA, CPENTA, CHEXA, CBAR, CROD, PSHELL, PBAR, PROD, PCOMP, PSOLID, RBE2/RBE3, MAT[n].) Users should also refer to the Thermal Desktop User's Manual (Reference 2) regarding which Nastran element types are recognized, and which types are deliberately altered during translation.

Armed with basic knowledge of both programs, analysts are advised to conduct a review of the Nastran bulk data set prior to translation in order to learn which specific element types are used and to identify potential translation problems. As will be discussed in this report, many of these problems are more easily resolved up front—by editing the bulk data file—rather than by editing the translated geometry within Thermal Desktop.

Analysts may also benefit from considering whether symmetry of geometry and environment permit adequate thermal characterization based on a section (e.g. half) of the original FEM, thus reducing labor and model size. Such decisions should be based on informed consultations with designers and other structural or thermal analysts. Ideally, the original model is truncated *before* translation via changes to the bulk data file; a modular organizational scheme facilitates this process. (Owners of the Nastran model are best equipped to perform the actual truncation at this point in the process.) Otherwise, unneeded sections can be deleted from the translated geometry within Thermal Desktop, although this method might be tedious and error-prone if the mesh is very fine.

Throughout this preliminary stage, analysts are encouraged to take advantage of ongoing consultations with mechanical designers and the stress analysts who developed the original model.

*Element Translation*

- Several of the Nastran element types that are recognized by Thermal Desktop represent elongated shapes (CBAR, CROD, and CONROD). Thermal Desktop automatically converts each of these "beam" elements into a linear SINDA conductor with one SINDA node on each end. The analyst should carefully consider whether the new configuration is significant and/or appropriate for a specific thermal model, then modify conductor and node definitions accordingly. In many cases, it is acceptable to delete all instances of reconfigured beam elements. However, the possibility exists that a beam represents a significant energy path, and thus should be represented with a user conductor or some alternate configuration within the thermal model.
- Thermal Desktop does *not* recognize several potentially significant Nastran element types. Because their omission may compromise the thermal model, analysts should carefully consider whether and how to represent their thermal effects.
    - CBEAM is an unrecognized element type, but has the same physical configuration as CBAR, CROD, and CONROD; indeed, all of the beam elements are *thermally* equivalent. If missing CBEAMs are needed for thermal reasons, they can be retained by changing them into CBARs in the bulk data file according to the formatting guidelines in the MSC.Nastran Quick Reference Guide (Reference 1).
    - So-called "Multi-Point Constraint" (MPC) elements—RBE1, RBE2, RBE3, RBAR, RSPLINE—and "spring" elements—CELAS1/CELAS2—are also unrecognized by Thermal Desktop. These "connective" elements impose *mechanical* constraints between planar elements, but their loss during translation results in a lack of *thermal* coupling. Such breaks between adjacent planar elements in the thermal network may not be readily visible; elements may appear to be thermally coupled when they are not, with obvious thermal implications. The *Show Free Edges* command in Thermal Desktop can be used to locate disconnected areas, and the *Merge Coincident Nodes* option is one way to reconnect them.
    - A CONM2 element, which is used in Nastran to represent a point mass with zero mechanical load, is *not* recognized by Thermal Desktop but may contribute thermally significant mass and/or surface area. Analysts should assess and, if necessary, compensate for the thermal effects of CONM2 elements.
- The Thermal Desktop translator does not recognize any Nastran bulk data entries in "large-field" format. A large-field entry occupies multiple text lines in the Nastran bulk data file; it is denoted by an asterisk appended to the end of the element type (e.g. CQUAD4*) on the first line, then another asterisk at the beginning of the second line. If access to Patran is available, the analyst select the small-field format option and re-output the bulk data file. Alternatively, the analyst can examine the bulk data file for any large-field entries and manually convert them to "small field" format according to guidelines in Reference 1.
- Experience has demonstrated the possibility of unpredictable behavior by isolated planar elements after translation. (Evidence suggests that this phenomenon may occur after the thermal analyst has modified the original mesh within MSC.Patran.) For example, a bad element may indicate an ambiguous *Active Side* definition for purposes of radiation modeling in Thermal Desktop: the active side(s) as indicated by the *Display Active Sides* tool does not agree with the element's thermal model definition. Discoloration of the element's surface in comparison to other elements may also indicate a bad element. The impact of bad elements on analysis results is unpredictable, so analysts are encouraged to visually inspect the model

and use the *Display Active Sides* tool to identify bad elements that may need to be redefined and/or re-imported in order to function properly.

*Materials and Properties*

- Generally, the translation of property values (e.g. density) from Nastran to Thermal Desktop is not reliable. Depending on how the material element is referenced by the Nastran model (e.g. via property elements PSHELL, PBAR, or PCOMP), newly created Thermal Desktop materials may take on zero or default property values after translation. The *Color by Property Value* function is useful for visually detecting incorrect values such as density and conductivity (of planar and solid elements only). The best way to solve discrepancies is to determine the correct material compositions by referring to mechanical design documentation and/or Nastran bulk data, and then create and assign the correct materials in Thermal Desktop.
- Thermal Desktop *does* recognize use of the Nastran property element PCOMP for composite planar elements; the translator creates and assigns the material name P#, where # is the Nastran PCOMP identifier (PID). However, thickness values of PCOMP elements are *not* correctly retained, and property values of the newly created materials are erroneous. The *Color by Property Value* function is useful for visually detecting incorrect thicknesses and property values, after which the analyst may need to calculate thermally effective thicknesses and/or properties and assign them accordingly.

*Geometry*

- Experience has shown that planar elements may unpredictably, and incorrectly, translate with *zero* thickness. The *Color by Property Value* function is useful for visually detecting incorrect surface thicknesses. Thermal analysts can consult with designers and/or stress analysts to identify correct element thicknesses, then edit the model accordingly.
- Due to the nature of Nastran modeling, translated geometry may contain areas where multiple planar elements are coincident in the same plane. This phenomenon is a product of techniques that are valid for mechanical stress analysis, but that might compromise a thermal model. Elements located in that same plane, but whose edges are offset, are detectable within the AutoCAD interface by relatively close inspection. However, *precisely* overlapping elements, whose edges line up perfectly, are virtually invisible; their coincidence is apparent when the user selects what looks like one planar element, but the AutoCAD Text Window reports the selection of two (or more) objects. Analysts should exercise great caution when deciding whether to retain coincident elements, because their inclusion in a single Radiation Analysis Group causes unpredictable and erroneous results during the ray tracing process.
- In order to achieve the desired orientation and location for the imported geometry in Thermal Desktop, it may be necessary to first re-orient and/or re-locate the meshed model within MSC.Patran. However, the process of creating new coordinate systems and/or translating between multiple coordinate systems within Patran may introduce confusion as to which system(s) will be referenced by the resulting bulk data set and by Thermal Desktop. (For example, the inclusion of a CORD[n] element within the bulk data file may have the effect of re-orienting or moving geometry upon importation into Thermal Desktop, so that

the imported grid locations may not match the grid locations as printed in the bulk data file.) Analysts are encouraged to verify location and orientation after importing geometry Thermal Desktop, consulting References 1 and 3 as necessary.

[References:

1. MSC.Nastran 2001 Quick Reference Guide, MSC.Software Corporation.
2. Thermal Desktop User's Manual Version 4.4, C&R Technologies, August 2001.
3. MSC.Patran 2001 Reference Guide, MSC.Software Corporation.]

*Summary of Useful Thermal Desktop Features*

The foregoing discussions have made mention of several Thermal Desktop tools that are useful for resolving translation problems. They are reiterated below.

- *Show Free Edges:* for locating uncoupled sections due to loss of elements during translation
- *Merge Coincident Nodes:* for connecting uncoupled sections
- *Display Active Sides:* for identifying bad elements that may need to be redefined and/or re-imported; for verifying consistent Active Side definition among adjacent elements
- *Color by Property Value*: for visually identifying translated values of thermal model parameters such as density and conductivity; for visually identifying translated element thicknesses

***Mesh with FEMAP***

Another method will be to bring the geometry into FEMAP, mesh it, and bring that mesh into TD.

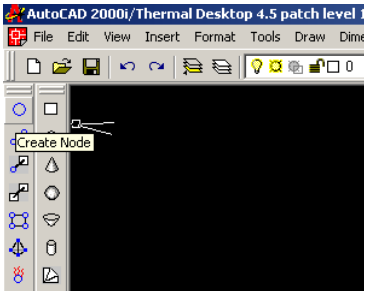
***Completing a thermal model and running it (see also "Short Guide to Thermal Desktop")***

Make layers for each component and the lump masses. Start with a file that already has basic material properties in it.

Select each component (by layer if possible), change to top/out active only (or as appropriate), add MLI, set thickness and material, set optical properties.

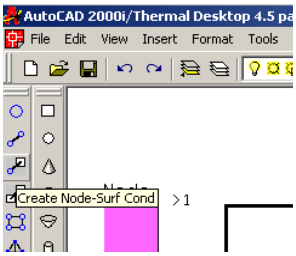
In order to add a mass node to each box (e.g., the electronics mass to be added to an electronics box), the easiest way is to make a separate layer (so that display of these nodes and their conductors can be easily suppressed). Then a node can be added for the additional mass in the box. This node will also be the easiest to add the power dissipation to. The node may be created in 3D space next to the box. If it is desired to have the node within the box, a line can be drawn within the box, such that the mid-point of that line can be selected when the node is created. Currently, no need for that is seen, so the node will be created external to the box.

To create a node: Thermal…FD/FEM…Node or select the Node button:



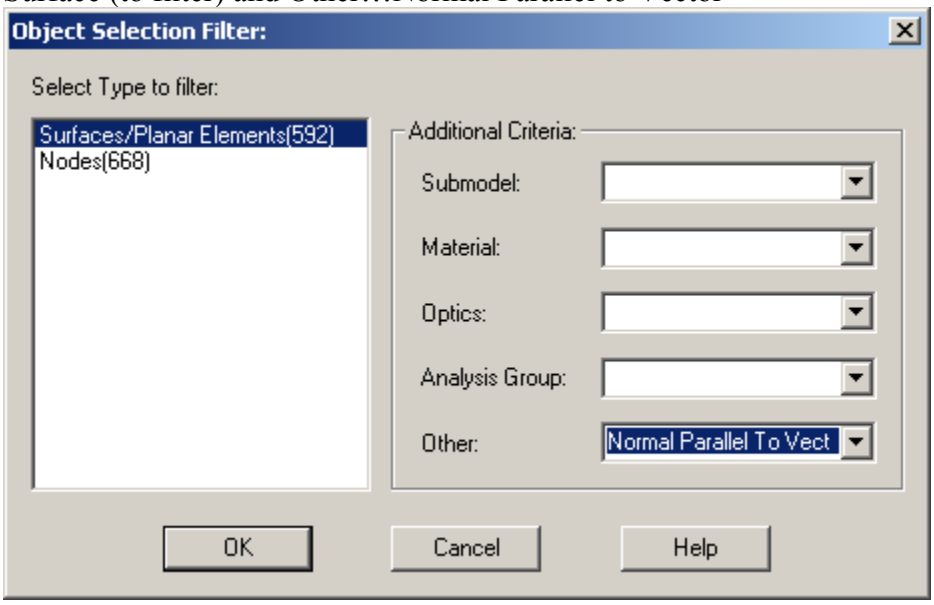

Then a mass (and power) can be assigned to the node, by doing Thermal…Edit (drag over the node to select it) for the mass, and the Thermal… FD/FEM…Heat Load on Nodes to assign a power.

You can attach the lumps to a box, by clicking the "Node-to-Surface" conductor:




If you want to select all or most of a box, the easiest way is to select to display only the layer the box is on.

You can select what sides of a box you want to choose by selecting a large volume, include the surfaces you want, and choosing Thermal…Edit. Then when the Filter screen comes up, choose Surface (to filter) and Other…Normal Parallel to Vector

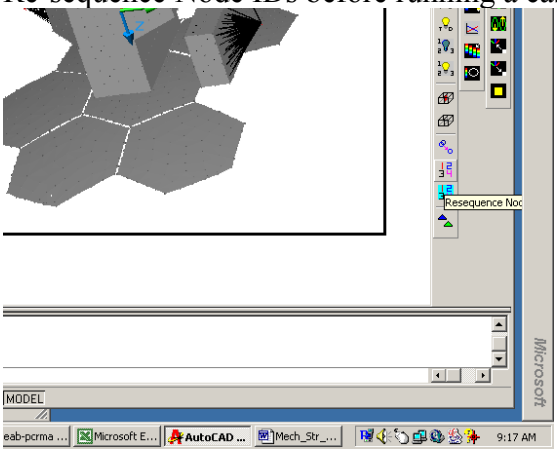



Then you can choose points to define the vector you wish to use.

Assign material and optical properties.

Assign Multi layer insulation (MLI) on surface (as insulation) by doing Thermal…Edit if necessary. (Create MLI as a material property by entering a non-zero value for the effective emissivity specified in the material property. The effective emissivity will be used to generate a SINDA/FLUINT radiation conductor between the surface nodes and the insulation nodes. If the material conductivity and surface thickness are set to non-zero values, a linear conductor between the nodes will be output. Normally you would set the thickness to zero so that only a radiation conductor would be generated. The actual insulation/MLI SINDA nodes are not represented graphically.) A special post processing check box can be used so that the user may select to display the calculated insulation node temperatures on the model (See "Post processing SINDA/FLUINT save files" on page 7-5 of the TD help).

Re-sequence Node IDs before running a case. Click on Resequence Node Ids button:




and then type “all”, <Return> twice.
Make sure you have all elements posted so that all actually get selected.
Make sure units are correct.
Make sure you are going to run in a clean directory (if necessary).
Do Thermal…Case Set Manager…define a case, and run it. You can run Transient or Steady-state (although steady-state will be somewhat meaningless for orbital), choose a default space node, etc.

You can view results and click through the time steps, as well as selecting as single node (while in post-processing view) and doing an X-Y plot on it.

## *Thermal Interface to CAD Using MSC.Patran Thermal*

*This describes the interface between the Mechanical CAD station running Pro/Engineer, the Thermal station running Patran Thermal, and the Structural station running Patran and NASTRAN.*

Import of geometry is much simpler in Patran Thermal – in general, simply import the .prt or .asm file directly from Pro/Engineer. (Do File…Import, and select Pro/Engineer format).

***General Tips for Importing and Meshing Pro/E Geometry in PATRAN***

When an assembly is imported, each part is brought in as a separate 'group' in PATRAN. This facilitates material and property definition by part, as well as meshing different parts separately. Parts that are unneeded for the analysis can be discarded by simply deleting or not meshing that group. For thin-walled or sheet metal parts, a surface geometry model can be built by importing the mid-plane geometry from Pro. With correct standards enforced for development of the Pro model, the Pro part file can be set up in layers, such that it is simple for the analyst to remove layers of unwanted detail, such as bolt holes and fillets, before importing the part (see here for a rough cut at Pro/Engineer standards).

Currently, only a tet mesh can be automatically generated on the imported Pro geometry of a solid part, when the part is a solid trimmed geometry (quads can be automatically generated on imported trimmed surfaces). To achieve an automatic hex mesh, the geometry must be re-created out of solids. When several parts are imported together in an assembly, the trimmed geometry is not treated as congruent between the surfaces of parts. Thus, the auto tet mesher will not generate a congruent mesh across the part boundary. For thermal analysis, this has the advantage that the analyst is forced to apply surface contact resistance between each part, and cannot ignore this physical effect. Thermal contact resistances are simple to apply as a boundary condition, over entire contacting surfaces or at desired points. For structural analysis the tet mesh not giving a congruent mesh across parts can cause difficulties. Depending on how the load path between parts is modeled, the tet mesh may require additional analyst labor and attention to the boundaries.

***Tips for Importing a ProE model into PATRAN***

- ***Try this first:***
  - Select "File", "Import", "Model", "Pro/ENGINEER" and let Pro/E set the tolerance. You can import either an assembly or a whole part.
  - If no luck, try doing the same, but do "Pro/E Options", "Import Preferences", turn the "Enable Tolerance Prompt" to "ON" (then say "NO" when it asks if you want Pro/E to reset the tolerance)
  - Try several global tolerance values to see if any bring in a good solid
- ***Try this if that does not import a solid:***
  - Attempt to create a b-rep solid and choose an imported model that has only a few "bad" seams between the imported surfaces. Do "Create", "Solid", "B-rep" and look for the gaps.

- Select "Geometry", "Edit", "Surface", "Sew" or "Edge-Match" to try and get the edges to match up. You can select all the surfaces for the entries in both boxes using Edge Match to try to sew all bad edges toegher at once. Also try surface edge matching with "Point" option enabled for surfaces that are not co-aligned.
- Look at the surface normals and make sure they are all pointed the same direction (outward).
- Try bringing in the IGES surfaces instead and do the Edge Match to create a Brep solid.
- If it is difficult to see the un-matching edges, try surface meshing with triangles, equivalence and look for free edges on the generated elements (Finite Elements, Show, Elements, Normals)
- Isolate only the bad part or section in of a model and try to create the solid with nothing else in the database.

- ***If all else fails:***
  - Try one of the following programs to clean up the geometry:
    - Pro/Desktop (external link)
    - CAD Fix (external link)
  - See this article (external link) for an interesting disucssion of the problems in picking up clean CAD geometry.
  - Try using the STEP format out of Pro/E and import into PATRAN (using separate module; may be able to get temporary licenses for a one-time use of the interface)
  - Send the ProE file and PATRAN database to MSC help line and beg for help
- ***Special Note:***
  - If a b-rep solid was created and it looks incorrect because the shaded image shows a hole or missing surface, try ignoring the hole/missing surface and mesh the solid geometry anyway. Then look at the shaded mesh to see if the flaw still exists. There is a bug in the shaded geometry image display section of PATRAN that sometimes neglects a surface in display, which makes it look like there is a hole where there isn't.

***<u>If you need a brick mesh on a white (B-rep) solid in PATRAN</u>***

- ***If the model is axisymmetric:***
  - Create a surface on plane within solid -- mesh a surface and rotate to build bricks
- ***If the model is extrudable:***
  - Mesh the surface and extrude (or sweep, rotate, etc.) to build bricks
- ***Then:***
  - Associate the FEM to the geometry afterward and check the associations to ensure that all of the bricks have been associated to the solid. In particular, do a color contour plot of any BCs applied on the surfaces of the solid to make sure they all apply correctly. If not, you can try changing your global model tolerance, or you will just have to apply a separate BC only on the FEM that was not picked up.
  - Delete the surface mesh when done.
- ***If the model is not amenable to the above, you will have to create a native PATRAN solid (blue solid) from the white (trimmed) solid in order to be able to directly auto-hex mesh it.***

***General Meshing Tips in PATRAN***

- ***You can only directly tet mesh a B-rep solid (white solid)***
- ***If the solid won't mesh:***
  - *Mesh the surface with triangles (if trying to build tetrahedrons) and then try volume meshing.*
  - *Try above after equivalencing the surface mesh before volume mesh.*
  - *Delete the surface mesh when the solid mesh is completed.*
- ***Or try:***
  - *Make sure no other mesh is in the database.*
  - *Try another global model tolerance (less or more) and global edge length*
  - *Try "Transform", "Solid", "Scale" (or Translate) and set to .999 or similar. Mesh the new solid and equivalence the new mesh, then remove the original solid.*
  - *Try moving the solid away from the rest of the geometry, mesh it, and then move it back (Group Translate option to get both geometry and mesh but both must be in a separate group)*
  - *Have Pro/E model made originally such that there is not direct contact between parts.*

For applying contact between touching parts, a good reference book for contact resistance values is the Genium Thermal and Fluid Data Book series.

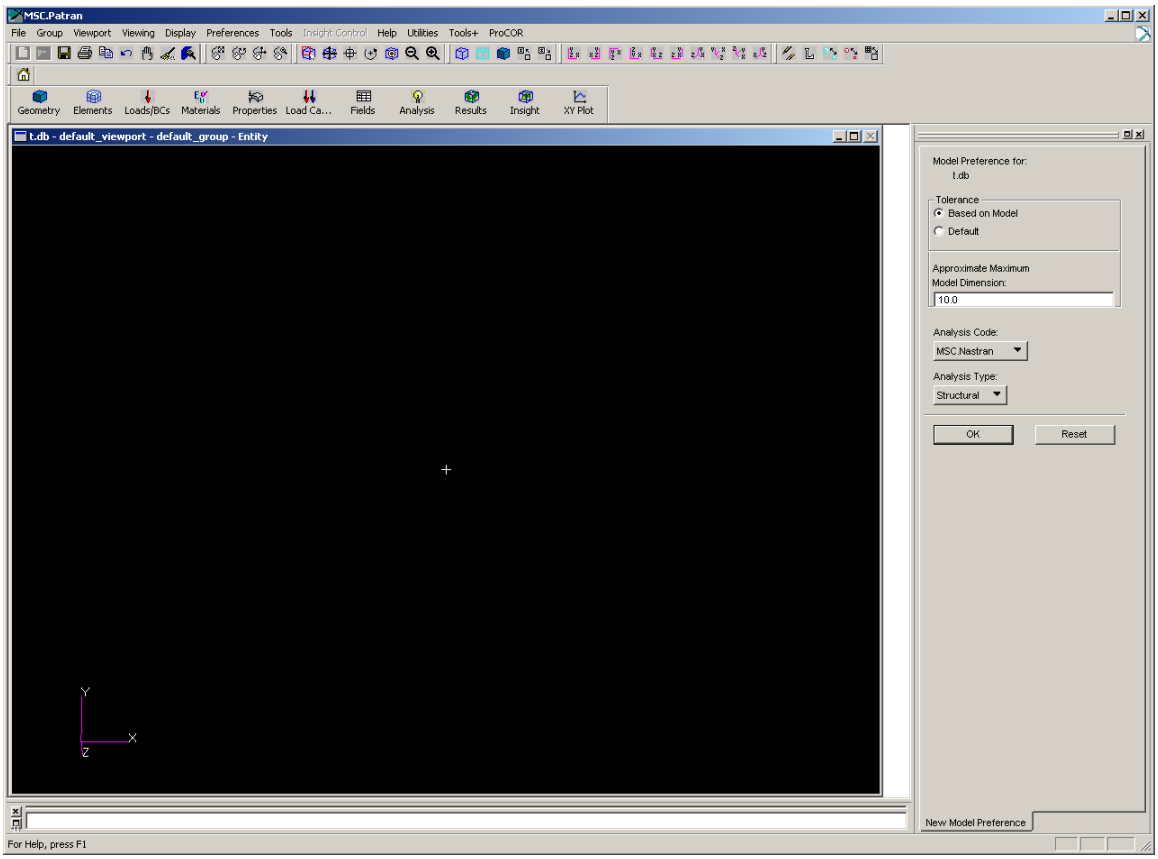
# Structural Interface with Mechanical/CAD Station

## *Determine Units and Coordinate Systems*

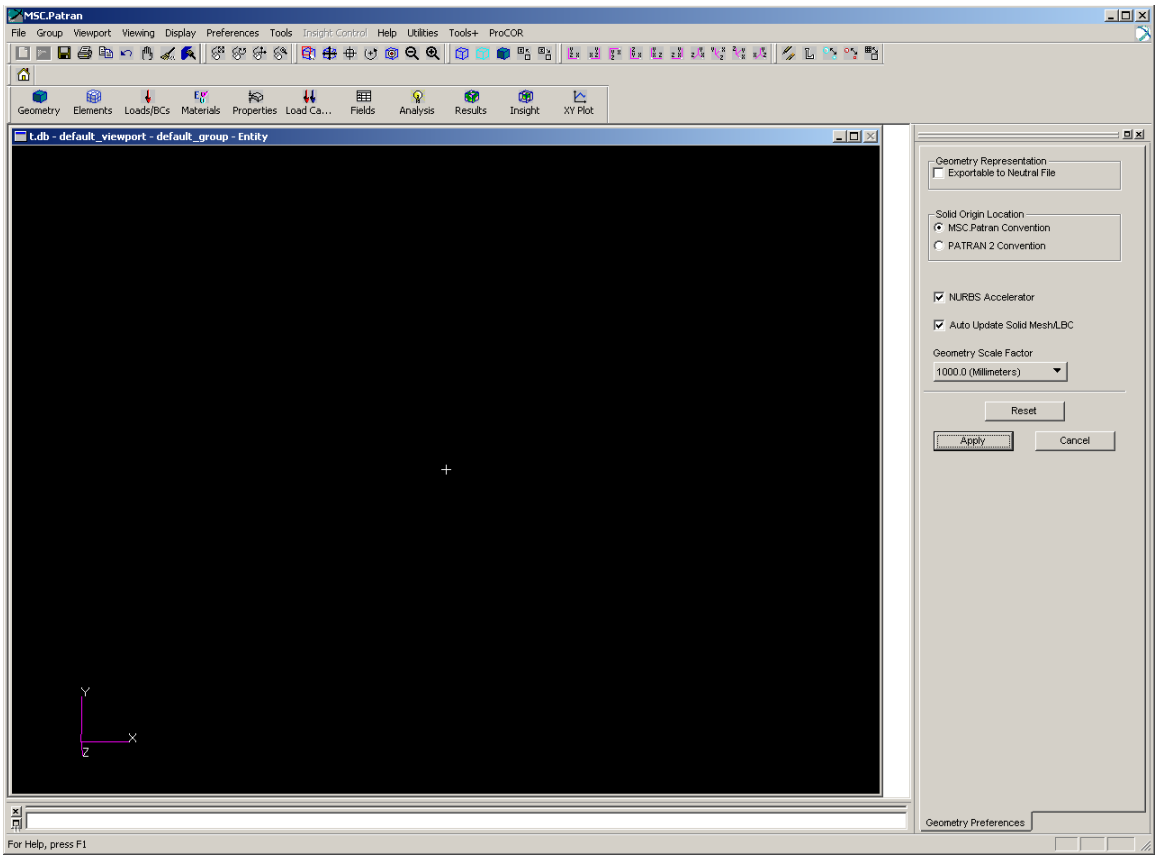
The Structural Station will get geometry from the Mechanical/CAD Station. The Structural Station should confirm the units and coordinate system used in the geometry. This information is important to set up the PATRAN database for the imported geometry.

## *Open new Database and Set Preferences*

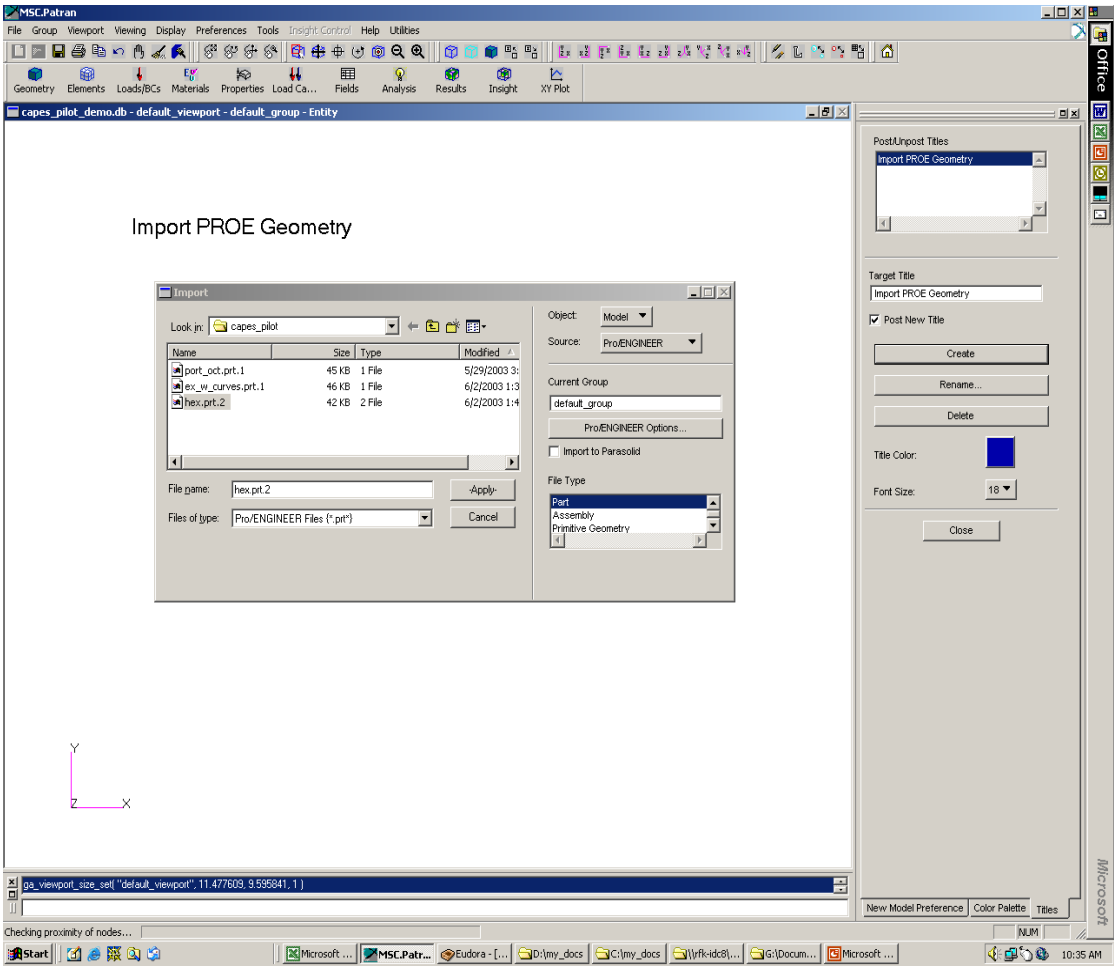
The Structural Station should confirm the units that the geometry is in and adjust the global tolerances if necessary. This should be done as soon as a new database has been opened. Use the top menu pick "Preferences, Global."
The "Analysis Code" should be set to NASTRAN and the "Analysis Type" to Structural.




If the units of the model are in mm, then geometric tolerances should be changed. This is done under the top menu tip "Preferences, Geometry."


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help Utilities Tools+ ProCOR
Geometry Elements Loads/BCs Materials Properties Load Ca... Fields Analysis Results Insight XY Plot
t.db - default_viewport - default_group - Entity
Geometry Representation
Exportable to Neutral File
Solid Origin Location
MSC.Patran Convention
PATRAN 2 Convention
NURBS Accelerator
Auto Update Solid Mesh/LBC
Geometry Scale Factor
1000.0 (Millimeters)
Reset
Apply
Cancel
Geometry Preferences
For Help, press F1

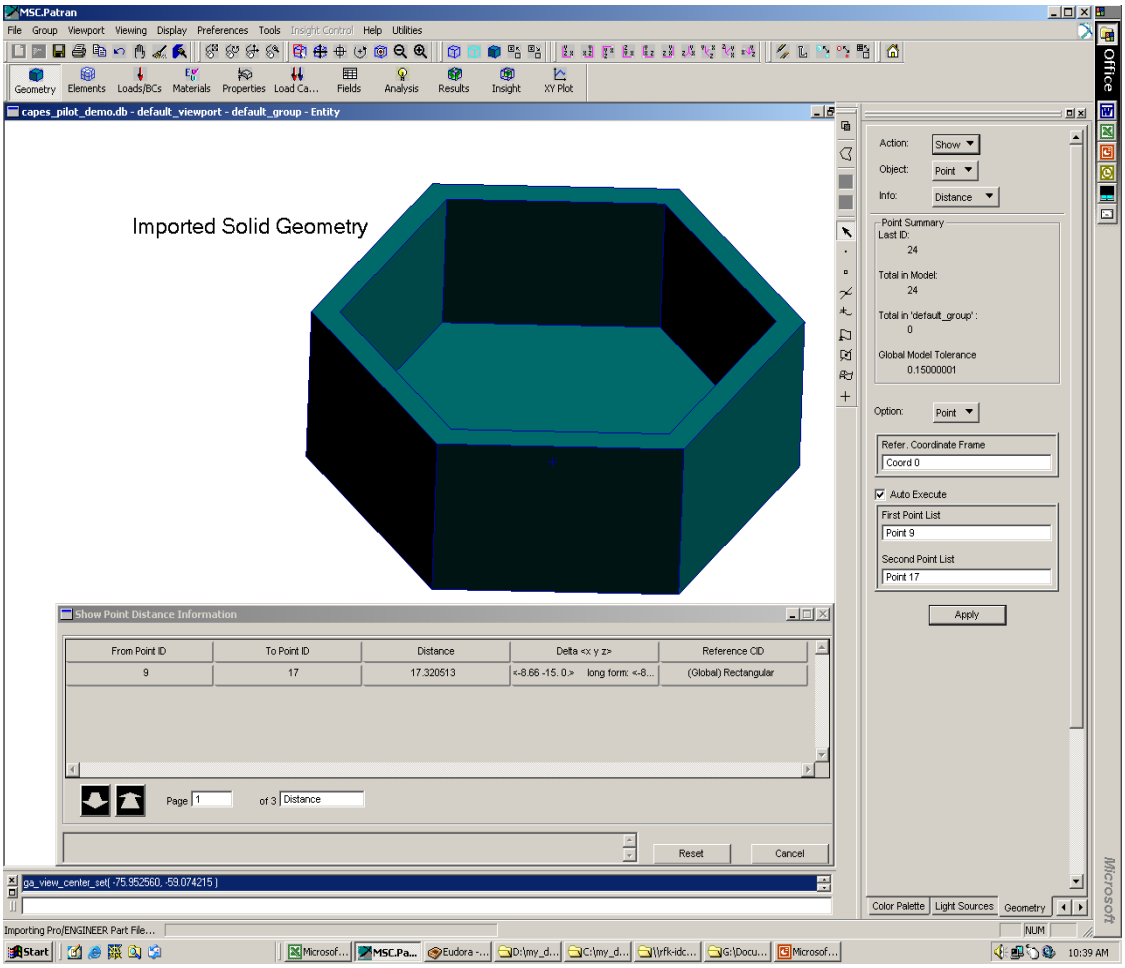
## *Interface between PROE Geometry and PATRAN*




PATRAN will import a PROE Part or Assembly. The example shown is for a part file. If the entire assembly is available, it should be imported. In order to determine that the current revisions of geometry are being used, PROE Intralink should be used to access the model and to place a working copy on the Structural Station. It is important for the Mechanical Station to use the same coordinate system and point of reference for all parts on an assembly. This will insure that when parts are updated, the new part (and not entire assembly) can be imported into the working PATRAN database without the geometry being "lost in space." The Structural station can use PROE Intralink to track current revisions of each part of an assembly.

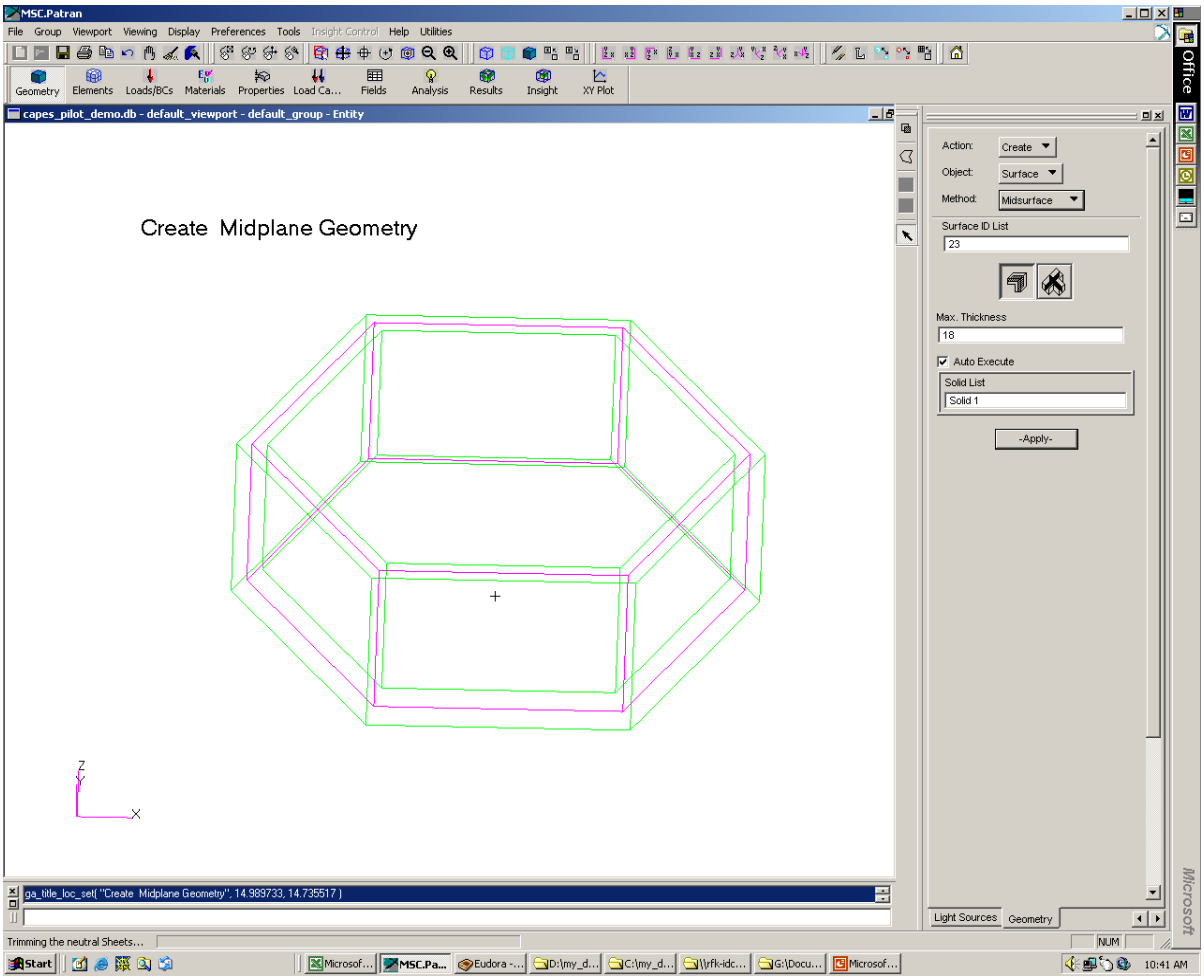
## *Examine Imported Geometry*

Examine the geometry to determine how many solids and surfaces have been imported. Confirm the units used in the geometry. In this example the units are mm. Determine the thickness of the solid walls. Use the "SHOW Point Distance" option in PATRAN. This information will be used to determine tolerances for the midplane generation and is also used as the thickness of the property card of a shell element.



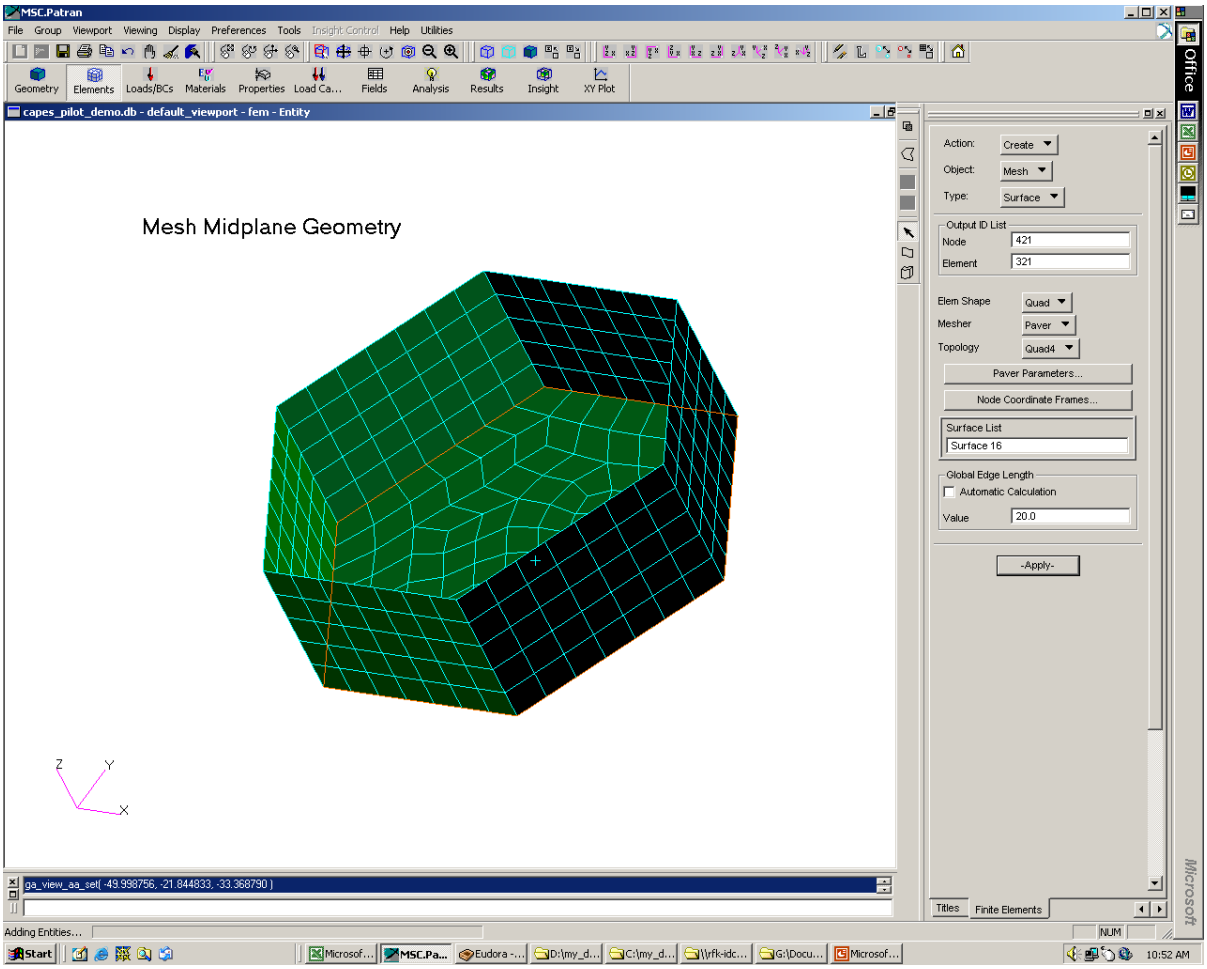

## *Create Mid-plane Geometry*

Create mid-plane geometry for each solid. There are two options for this procedure, using automatic or manual. For simple geometry, use the automatic option to create all surfaces in one step. It is important to choose an accurate value for the "Max Thickness" parameter on the PATRAN form. The newly created surface will show up as magenta lines. If working with mm, it may also be necessary to change the global tolerance.




## *Mesh Surfaces with Paver Option*

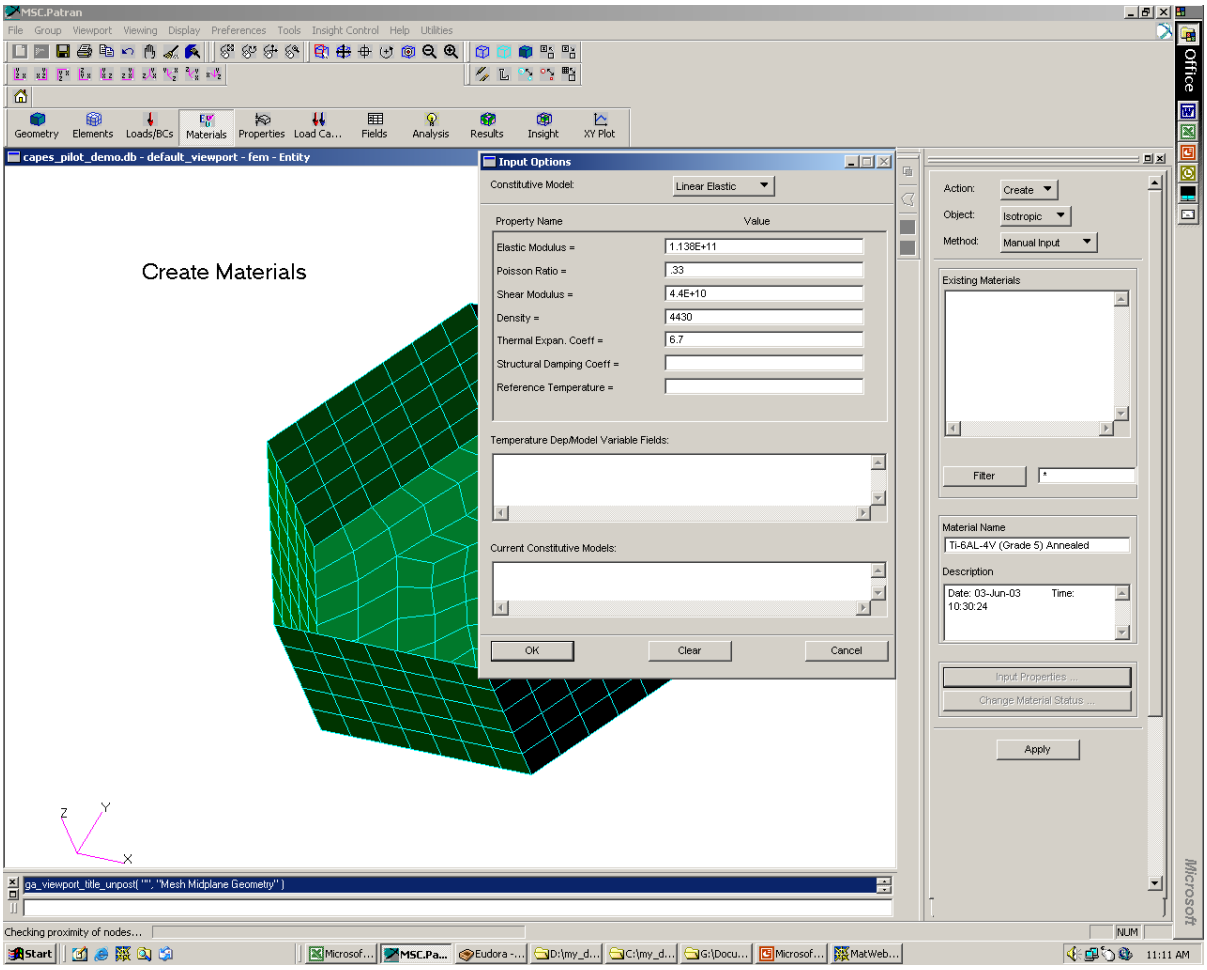
Once you have midplane surfaces then you can easily mesh with the paver option.



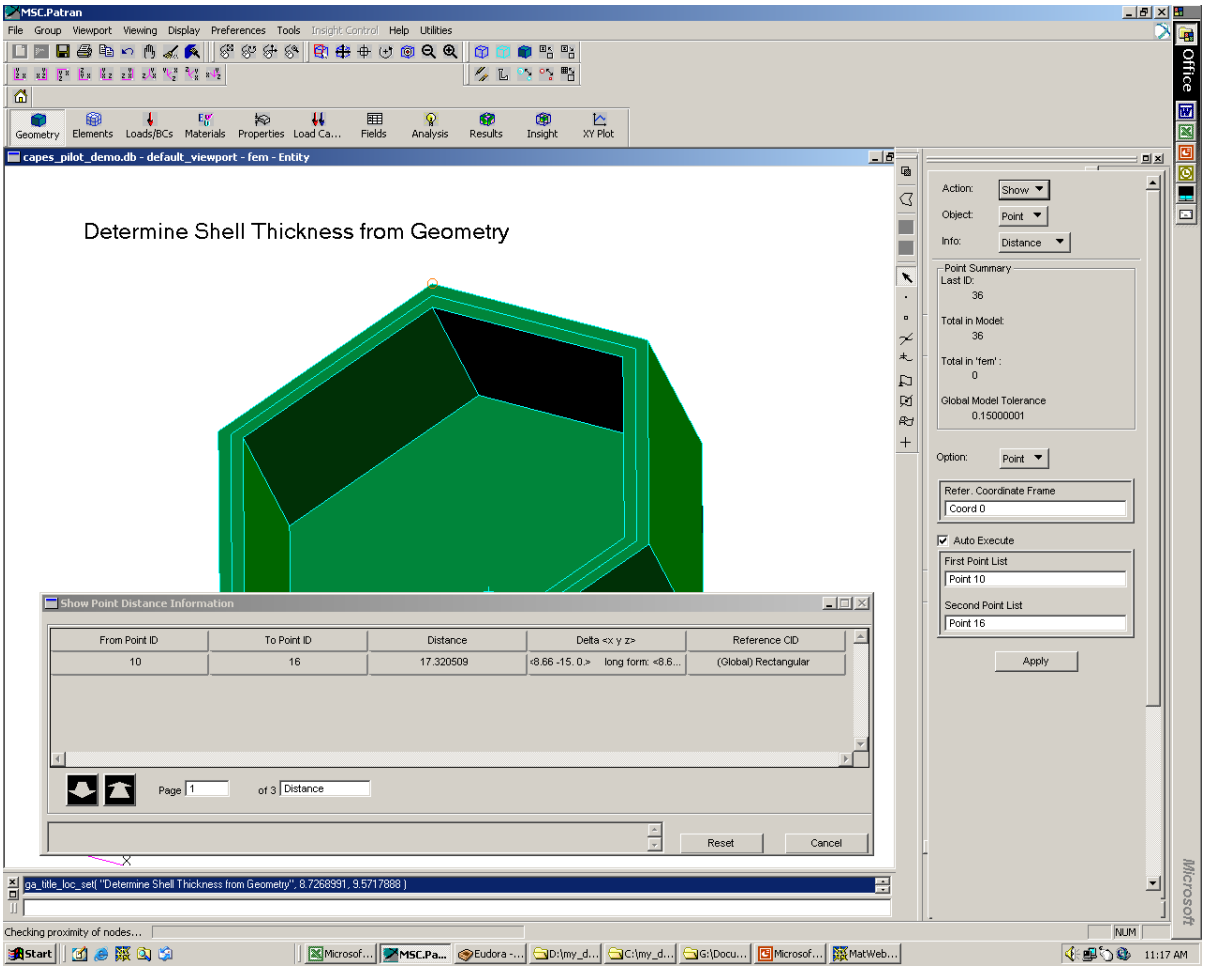

## *Create Material*

Material type used in the model is available directly from the PROE Intralink listing. PATRAN Material Selector can be used for automatic input of material properties of common materials. Properties of composite materials that may likely be used in the project can be researched in advance.

This next picture shows the material data input form. "Manual Input" can be used as the method choice or "Material Selector" chosen for automatic material data with the correct units. Shell element thickness is determined from geometry.


MSC.Patran
Create Materials
capes_pilot_demo.db - default_viewport - fem - Entity
Input Options
Constitutive Model:
Linear Elastic
Property Name
Value
Elastic Modulus =
1.138E+11
Poisson Ratio =
.33
Shear Modulus =
4.4E+10
Density =
4430
Thermal Expan. Coeff =
6.7
Structural Damping Coeff =
Reference Temperature =
Temperature Dep/Model Variable Fields:
Current Constitutive Models:
OK
Clear
Cancel
Action:
Create
Object:
Isotropic
Method:
Manual Input
Existing Materials
Filter
Material Name
Ti-6AL-4V (Grade 5) Annealed
Description
Date: 03-Jun-03
Time:
10:30:24
Input Properties ...
Change Material Status ...
Apply
Checking proximity of nodes...
NUM
11:11 AM

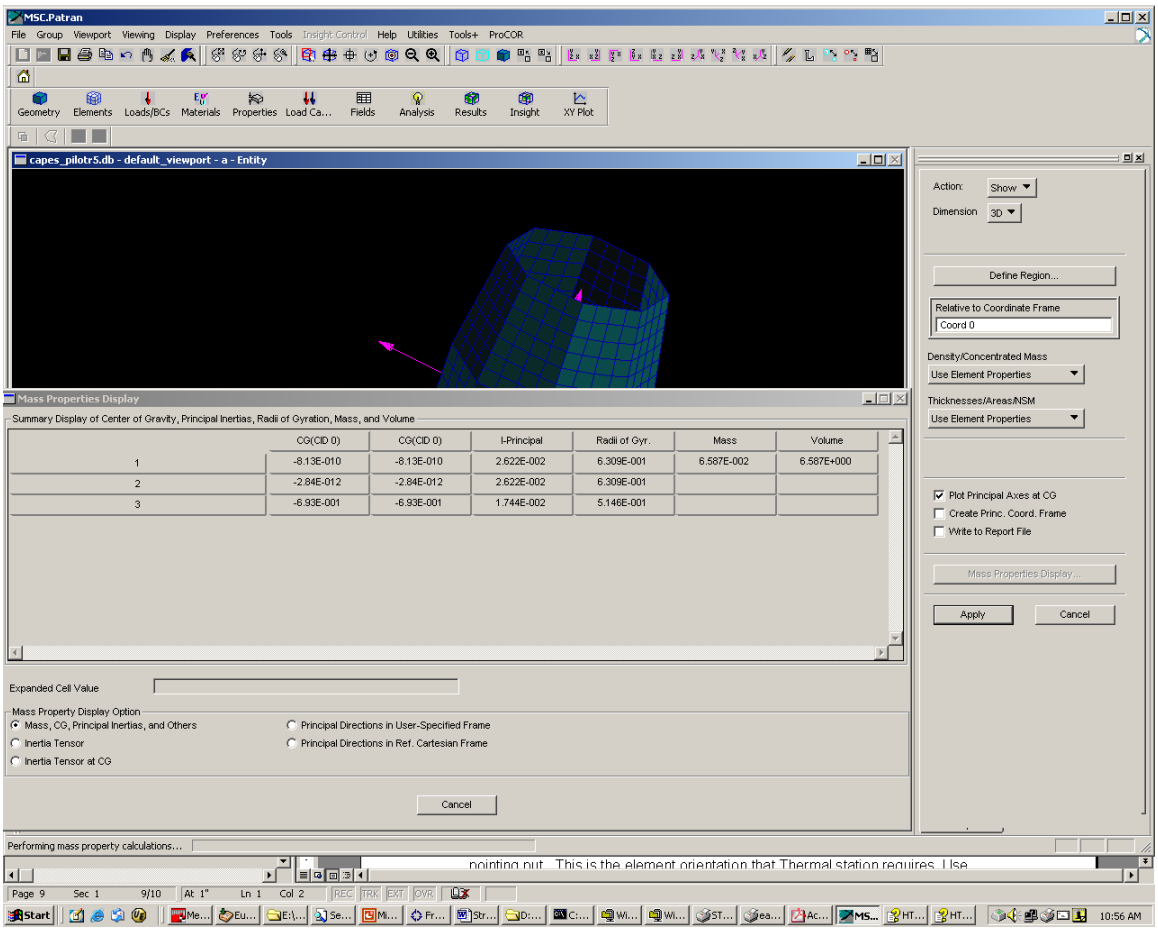
## *Determine Shell Thickness from Geometry*




Determine the shell thickness from the solid geometry and create the PSHELL property data records.

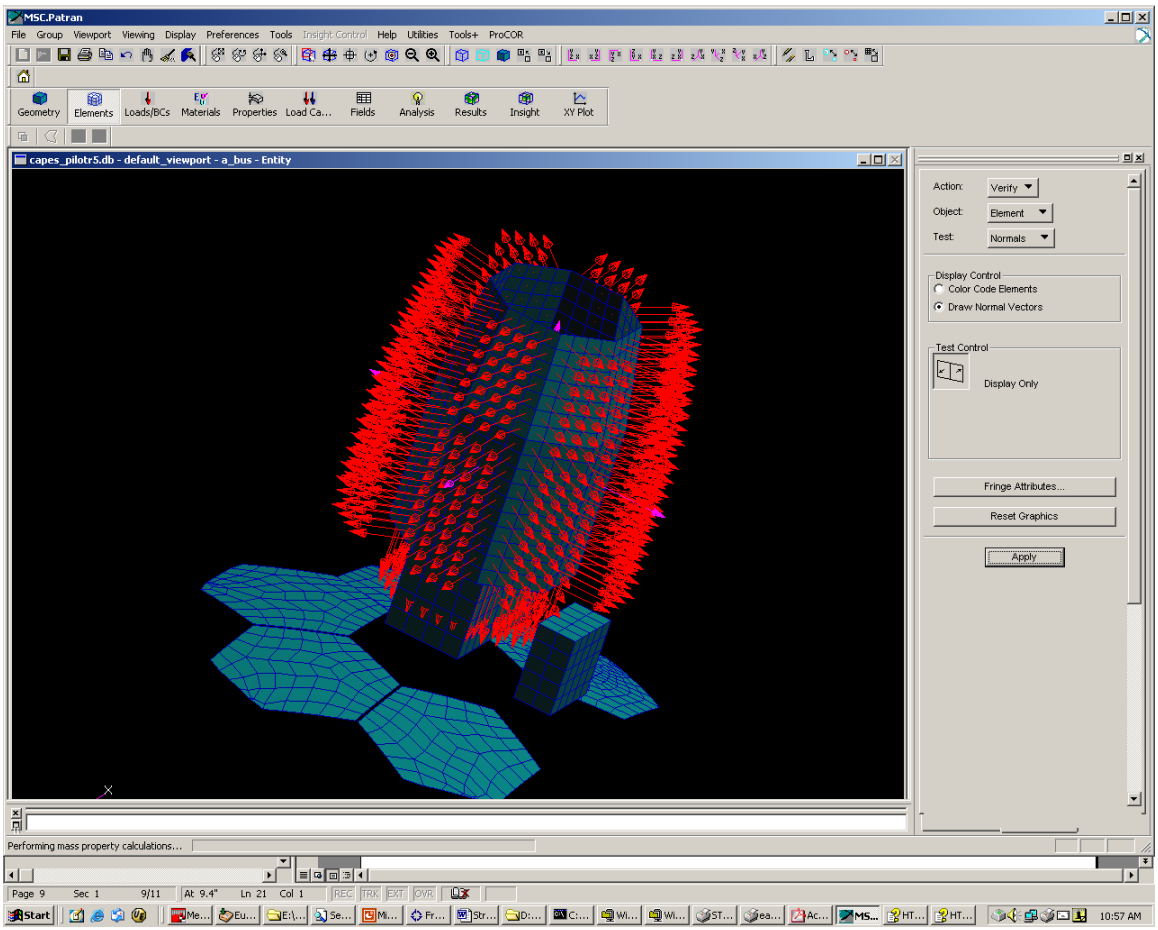
## *Determine Mass and CG information for each component.*

Once the material properties and thickness have been assigned, use the “Mass properties” calculation under the tools menu to determine a weight and cg. Compare this information to the data from the CAD station.




## *Check Shell Normal Directions*

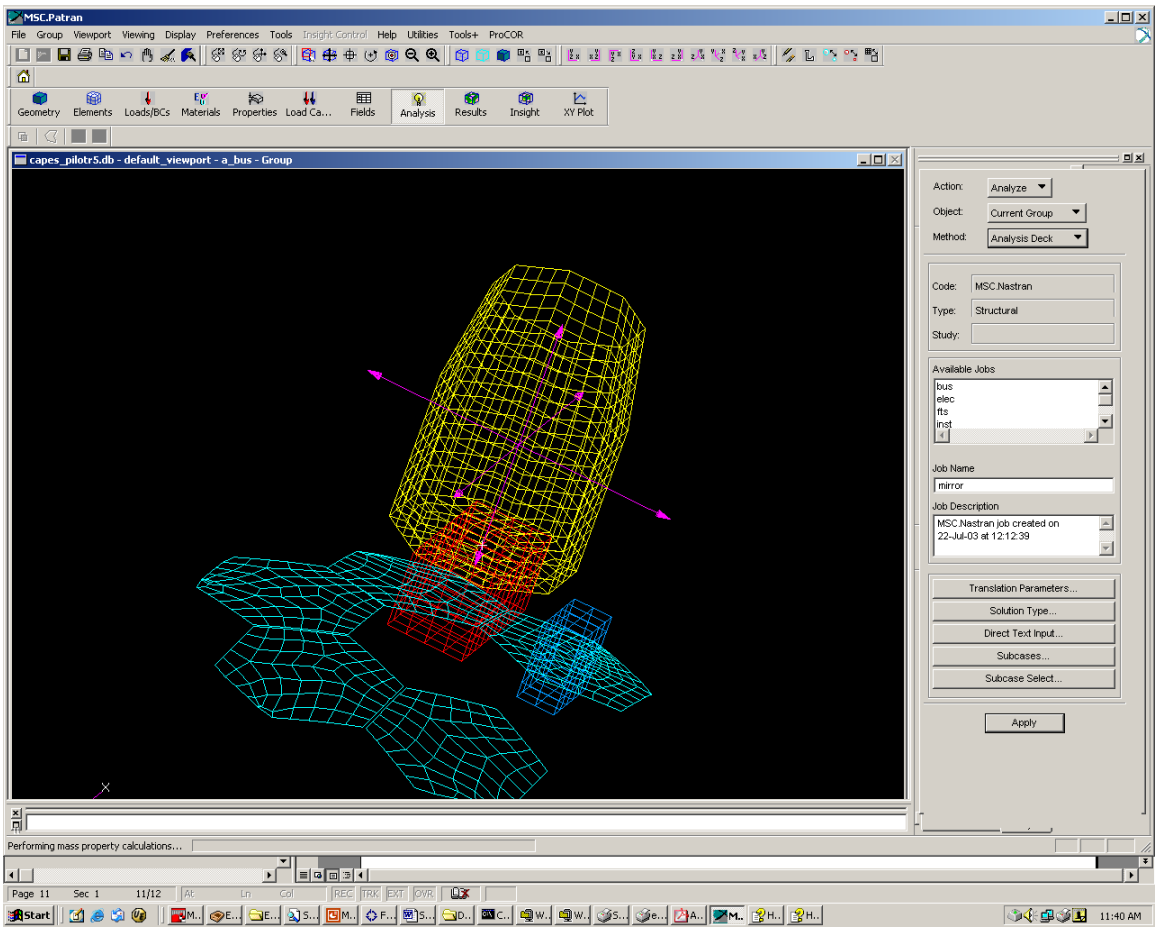
Check shell normal directions for each component in order to verify that they are pointing put. This is the element orientation that the Thermal station requires. Use the edit element reverse option if necessary.




# Structural Interface with Thermal Station

## *Output Bulk Data File for Thermal Station*

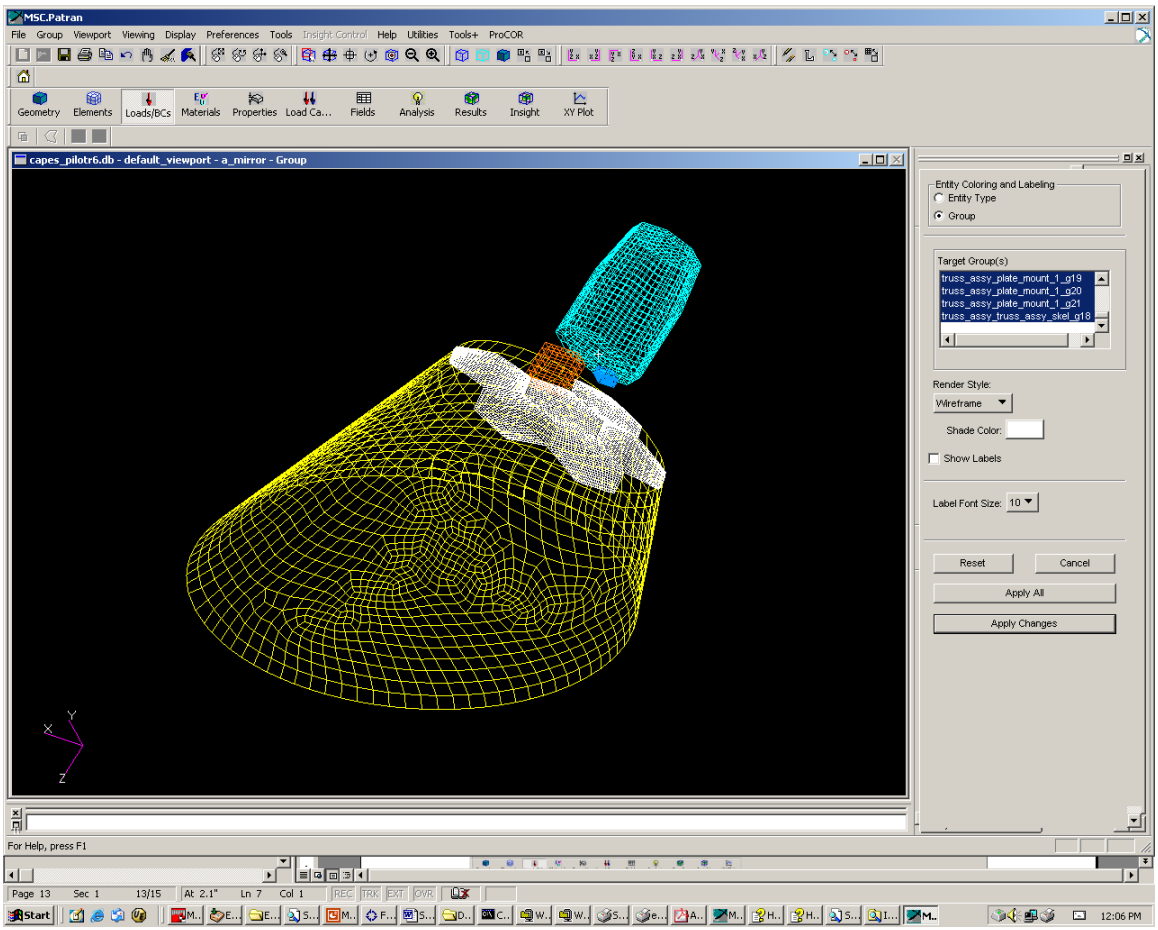
Create a separate group for each component of the structure. Under the "Analysis" menu, select "Analyze, Current Group, Analysis Deck" to create a NASTRAN bulk data file. There should be separate bulk data file for each component placed in the directory for the Thermal Station.




This picture shows the separate components color coded.

## *Update the Finite Element Model*

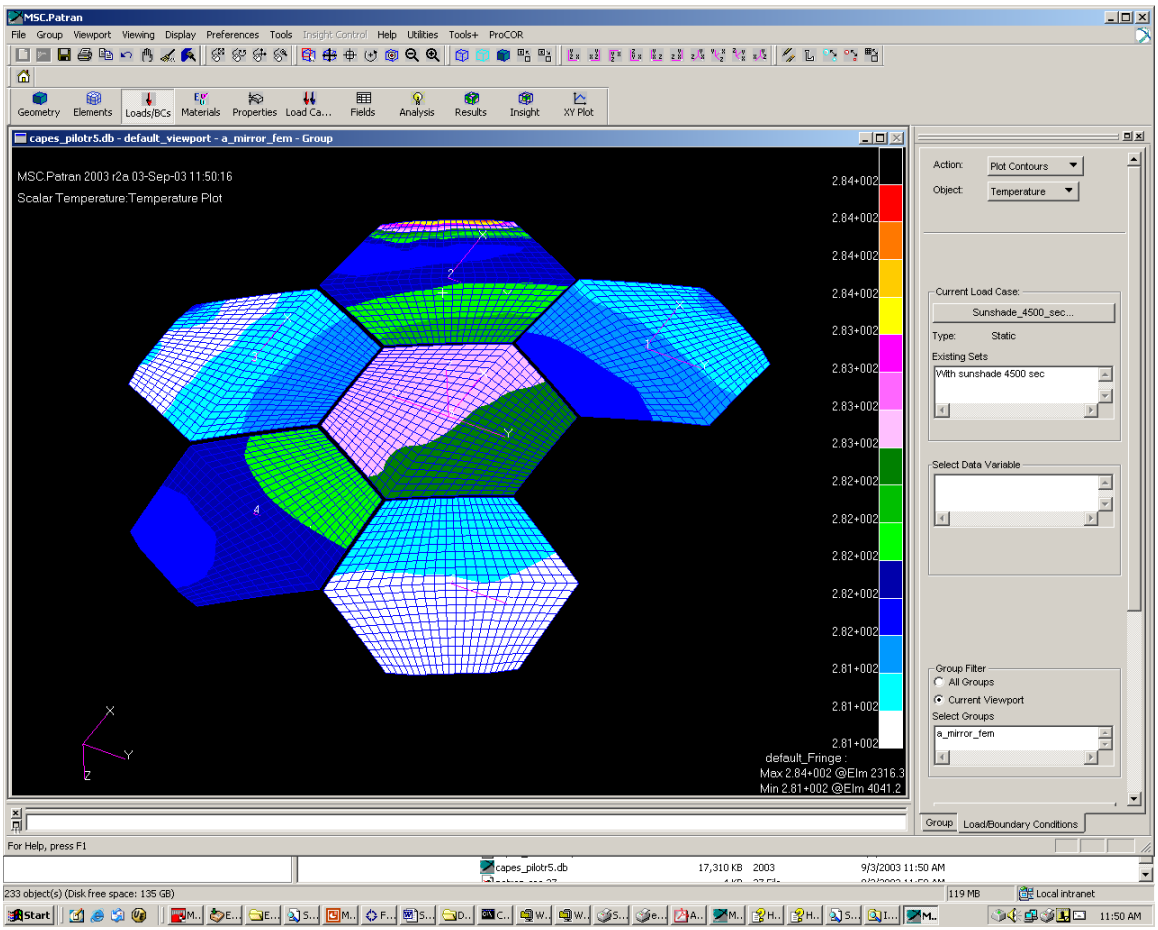
During the design process, addition geometry will become available from the CAD station. PROE Intralink should be checked for updates and new components should be imported. During the CAPES project, the sunshade was designed after the initial bus and mirror geometry and was an important for thermal analysis.




## *Import Nodal Results File from Thermal Station (Thermal Desktop)*

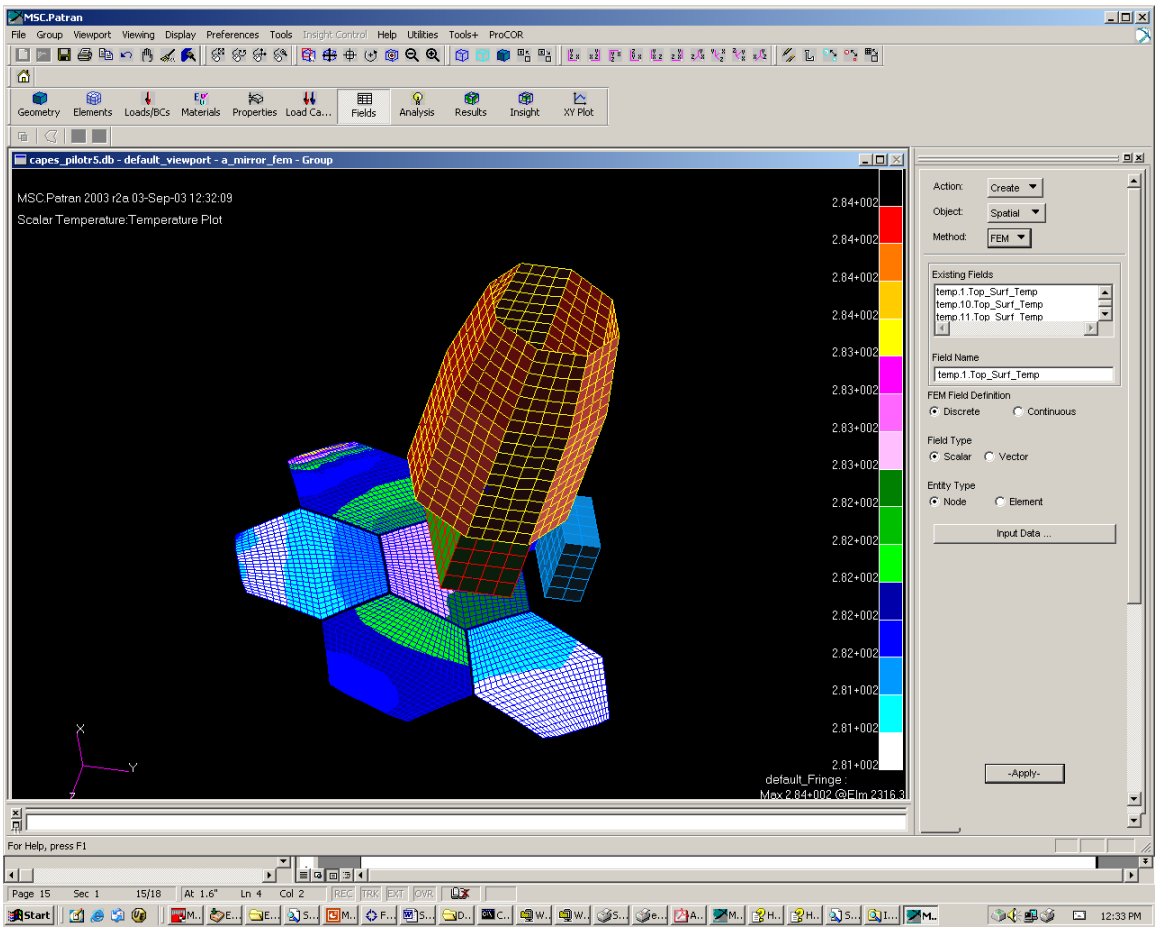
When thermal results are available from the thermal station, import them into PATRAN and make plots of each nodal results file.
To do this:
In Patran, NASTRAN preference, do File…Import. Select Object: Model, Source: MSC.Nastran Input. In File name box, change to *.dat (this is the file type created by Thermal Desktop). Select the file desired. The TD results will be imported as a Load/BC in Patran, named temp.N. To view, go to Loads/BCs and do Action: Plot Contours, Object: Temperature, and select the set that was created on import.

It is helpful to place these plots on the thermal station in order for the thermal analyst to examine them for correct interpolation, and for the thermal analyst to also place plots corresponding to each nodal result file in the directory that is used to transfer data to the structural station.



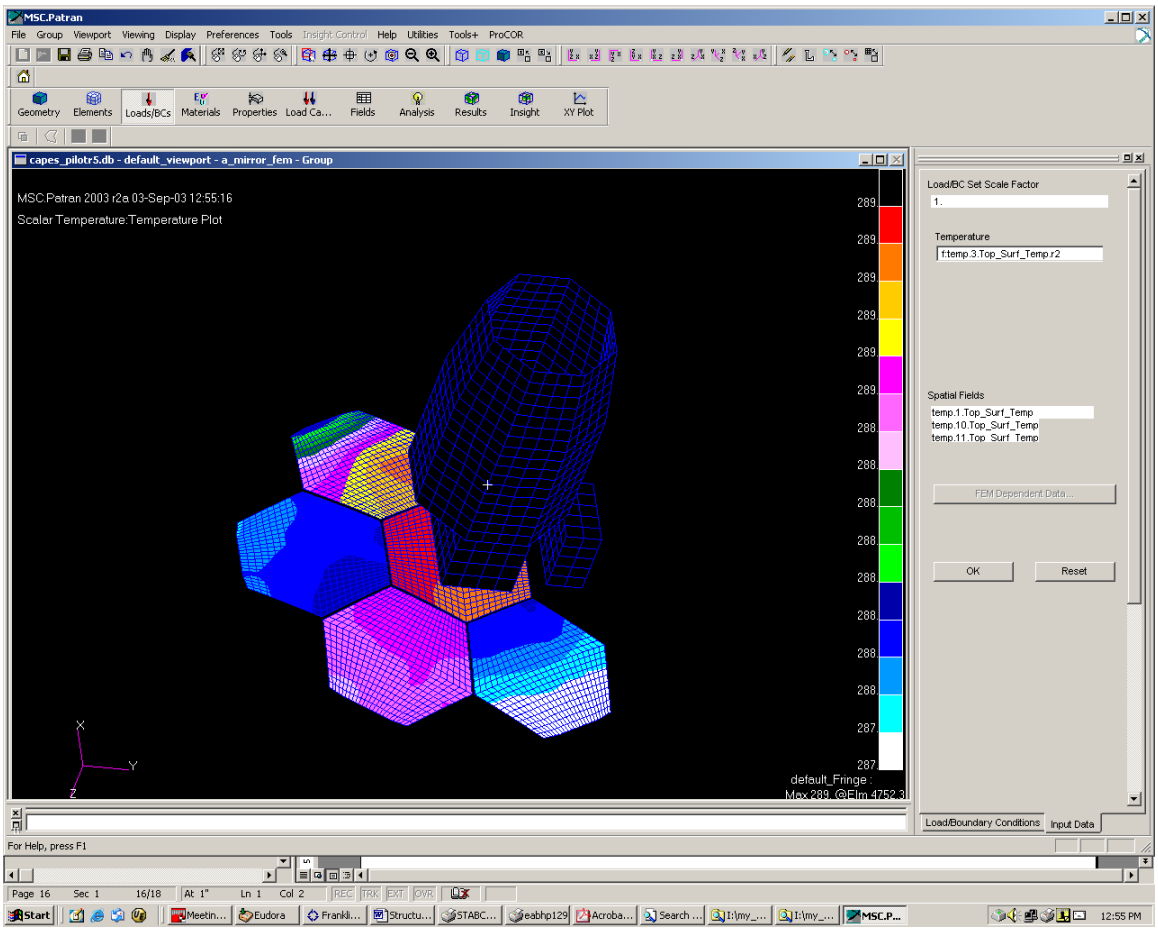

## *Create Fields*

Use the result contour plots to create fields for each set of thermal data.



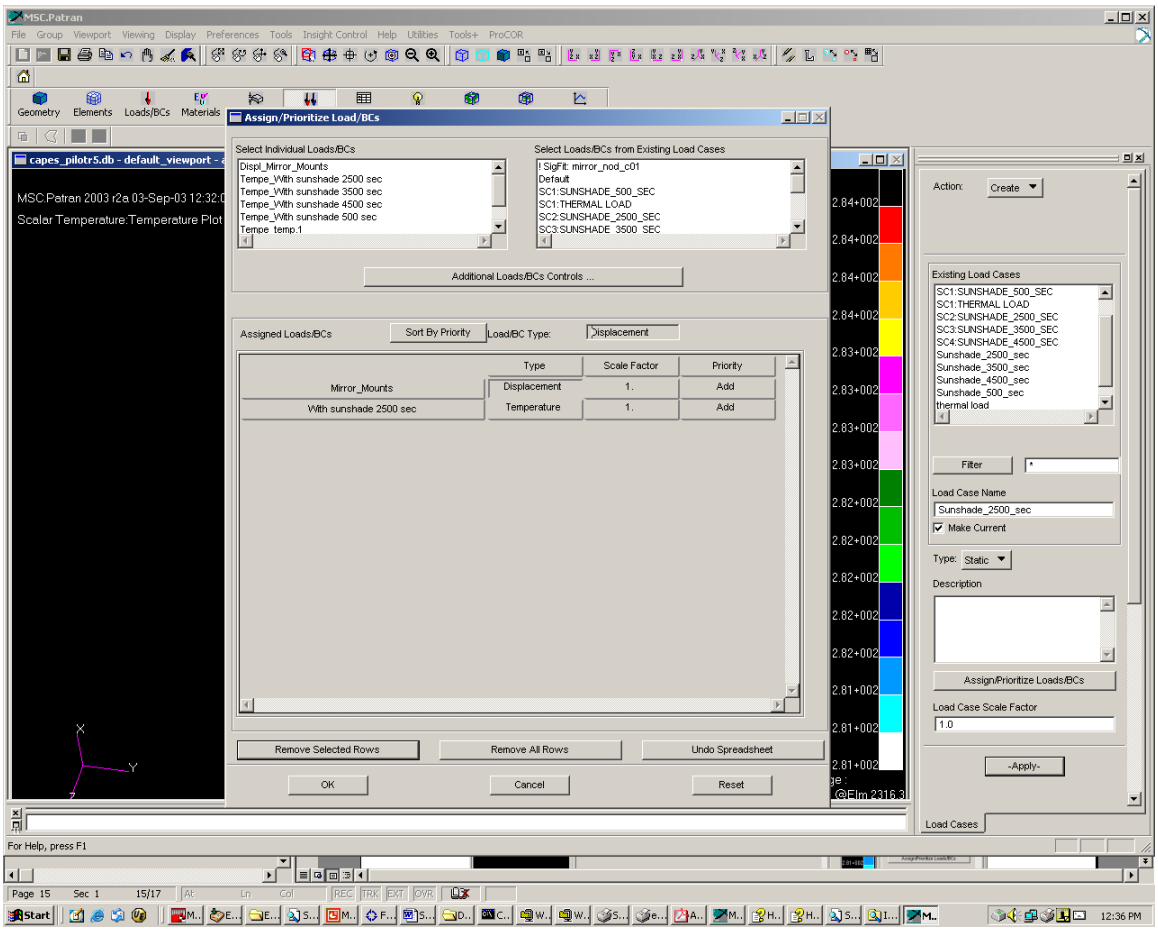

## *Create Thermal Load Sets*

Use the fields to create a temperature load for each case.



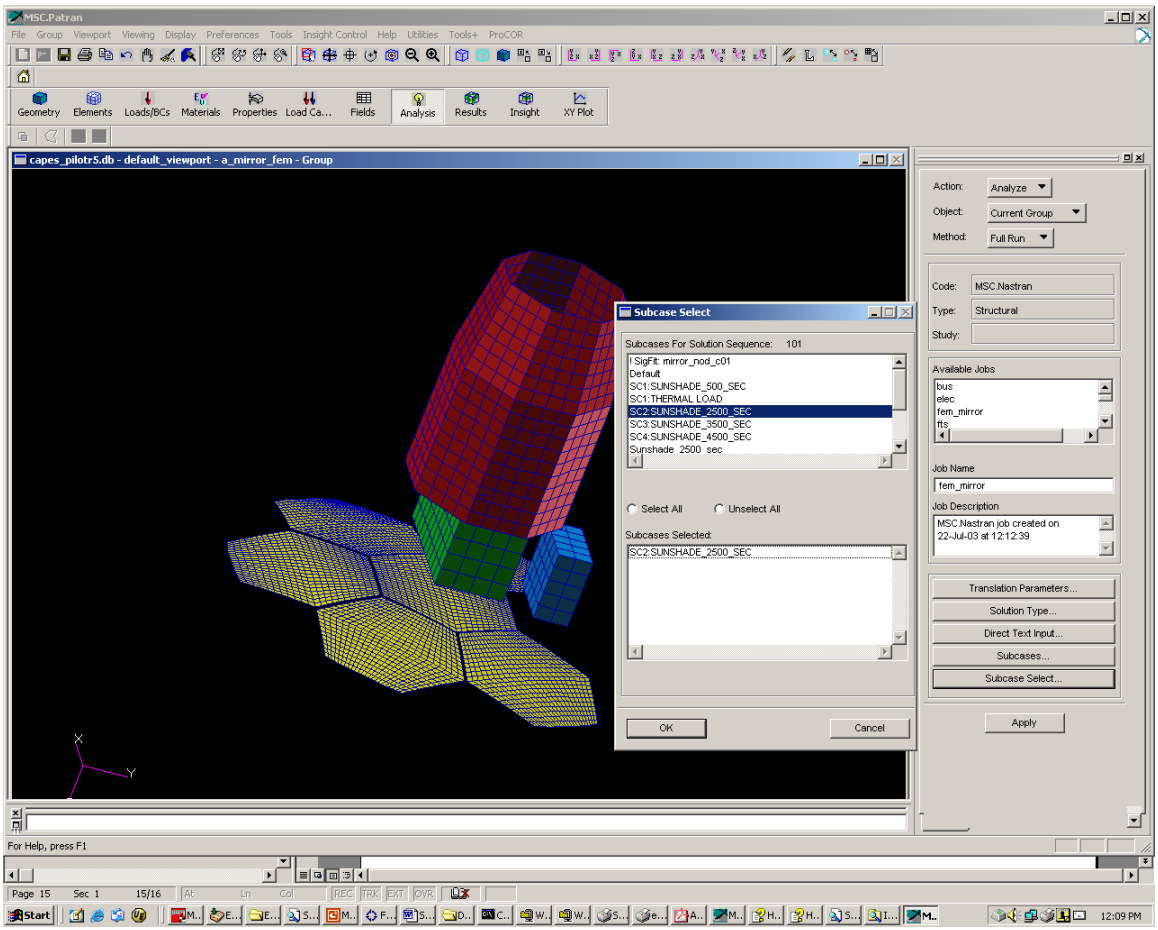

## *Create Load Cases*

Create load cases for each thermal case. In this example, a load case is created for a thermal condition at 2500 sec with a sunshade. The boundary conditions are assumed to be the approximate mirror mount locations. The case will be used to provide data to the optical station.




## *Set up NASTRAN Analysis and Run Job*

Set up analysis and run NASTRAN in order to calculate stress and deformations. In this example, only the mirror is being analyzed. Deformations will be calculated to use as input into the SIGFIT program. SIGFIT will calculate the Zernike polynomials required by ZEMAX (the optical code used by the Optical Station)




## *Import Nodal Results from Patran Thermal*

In the Hyper-X program, once the thermal analyst completes the transient thermal analyses for a given component, temperatures for each time step are provided to the structural analyst in the form of "nodal results files." These nodal results files (nrf) are binary files containing node and temperatures data from the thermal analyses. The files are not directly usable by the structural analyst because the structural and thermal models are normally, if not always, different (different mesh densities, for example). Therefore an interpolation procedure must be run in order to transfer the thermal results onto the structural model. An outline of the steps involved is given below:

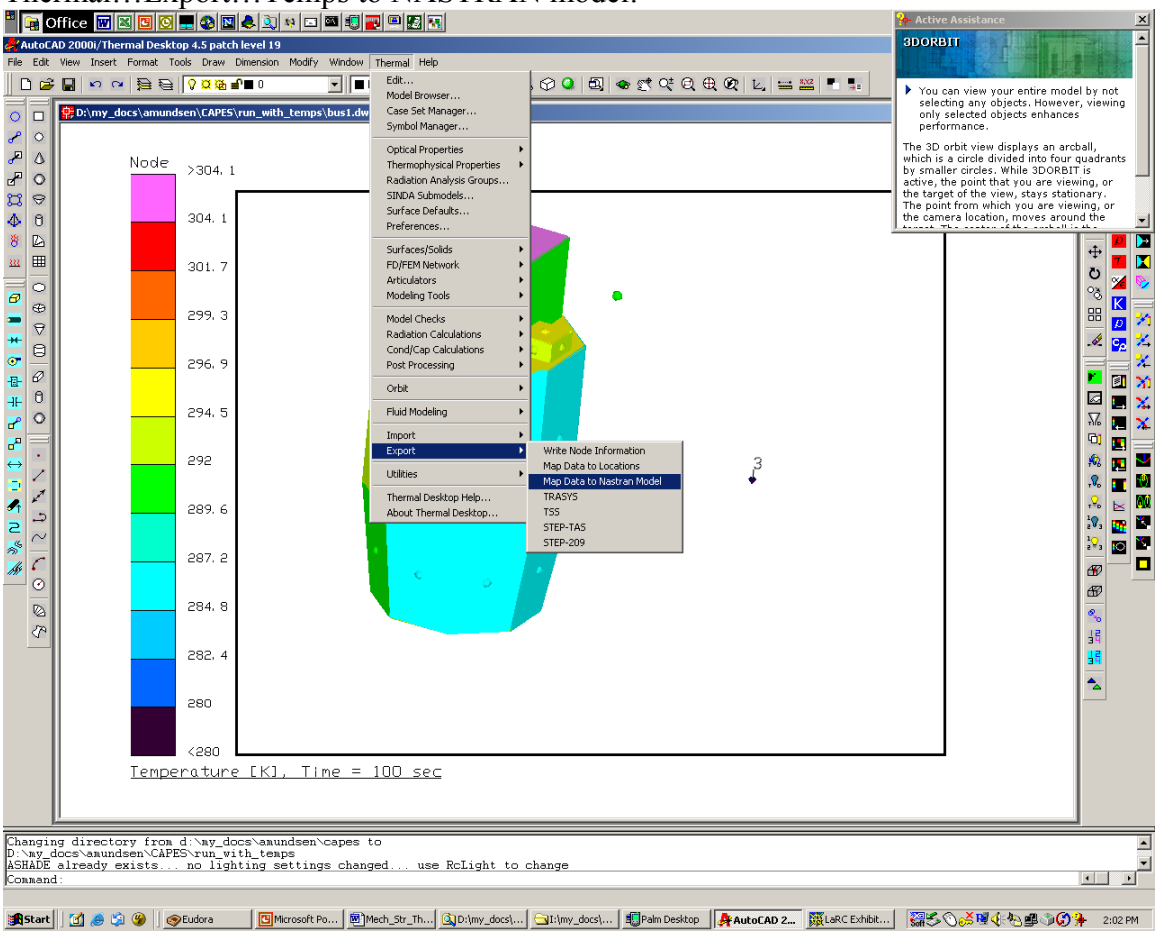
1. Obtain nodal results files from the thermal analysis.
2. Obtain a Patran thermal neutral file describing the nodes and elements of the thermal model.
3. Generate a Patran structural neutral file describing the nodes and elements of the structural model.
4. Execute the Patran Patq utility to interpolate the temperatures from the thermal model to the structural model. Inputs are the thermal and structural neutral files and the nodal results file. Output is a Patran neutral file containing the interpolated temperatures at each of the structural nodes (one output file for each nodal results input file). Model Center can drive this step.

5. Either import the structural temperature neutral files into Patran and create load cases corresponding to each thermal analysis time step, or write a translator to convert the temperature neutral files into Nastran bulk data TEMP entries to include in a Nastran bulk data file of the model. Model Center can drive this step.
6. Generate Nastran files for each load case and execute Nastran. Model Center can drive this step.
7. Either import Nastran results into Patran and evaluate structural responses for each time step to determine the worst case, or use Model Center to parse the Nastran results to find and display the worst case.

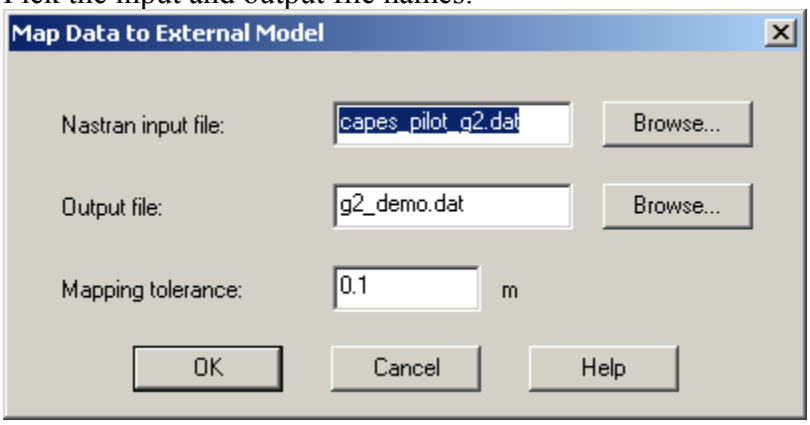
# Thermal Interface with Structural Station

## *Exporting Temperatures from Thermal Desktop to NASTRAN model*

*For this to work, the two geometries MUST be in the same locations and have the same units*
Thermal…Preferences to set model to same units as NASTRAN model (e.g., mm) (uncheck "Do not rescale model to new units")
(Outside, within Windows, change NASTRAN node file extension to .dat if necessary)
NASTRAN model must have BEGIN BULK card, can have all info as well (temps, header, etc)
Thermal…Export…Temps to NASTRAN model:



Pick the input and output file names:




Remember to set back to desired units if necessary (e.g., m)
This will map all parts of the model, not just what's displayed, so if you have a close part that is a different temperature, it can affect the temperature output map. A quick work-around is to create a dummy model file, and delete the offending part before making the mapped files.

Make an image file (alt-Print-Screen button) and put it in the same folder so that structural person can use it to check how the temperatures came across. It may be best to set up a single defined spectrum range so that all plots use the same spectrum and can be more easily compared to the structural plots.

## *Exporting Temperature Results to Structural (using Patran)*

Create results (nrf) files for the cases to be evaluated by the structural analyst.
Create a neutral file of nodes and elements only. May need to create multiple files if there are elements that are very close to each other with very different temperatures.
Transfer to structural analyst for import as in section "Import Nodal Results from Patran Thermal"

## *Importing Thermal Desktop model from NASTRAN*

As described above, in summary:

Set model to correct units for imported geometry (e.g. mm)
Thermal…Import…NASTRAN
Looks for .bdf file (may have to chg file extension)
Set model back to desired units

## *Importing Patran Thermal model from Patran structural model (NASTRAN)*

A complex assembly of Pro/Engineer parts, or a single part, can be imported directly into MSC/PATRAN. The part is then meshed; structural analysis is performed using MSC/NASTRAN, and thermal analysis is performed using either PATRAN Thermal, with P/Viewfactor or TRASYS for radiation. These analyses can be performed from the same PATRAN mesh or different meshes. If the analyses are done using different meshes, it is still possible to derive temperature-driven structural deflections using interpolation of thermal results from one mesh to the other.

If the model is to be used by both the structural and thermal analyst, communication before development of the model is extremely important. The analysts should discuss:

- analysis code(s) to be used
- whether a record of the node and element numbers need to be kept for the thermal analyst (and whether the numbers need to be sequential in either set)
- level of detail that needs to be retained from Pro/E model (bolt holes, etc.)
- appropriate level of mesh detail (and variation)
- whether to use solids or plates for thin-walled parts
- for radiation, which way surface normals should face
- for structural analyst, control and standardization of x, y and z directions of individual elements
- how to minimize information which must be deleted by the thermal analyst
- how temperatures will be imported to structural model (and whether same mesh will be used)
- units (dimension and temperature) -- note: dimensional units must also be worked with the designer

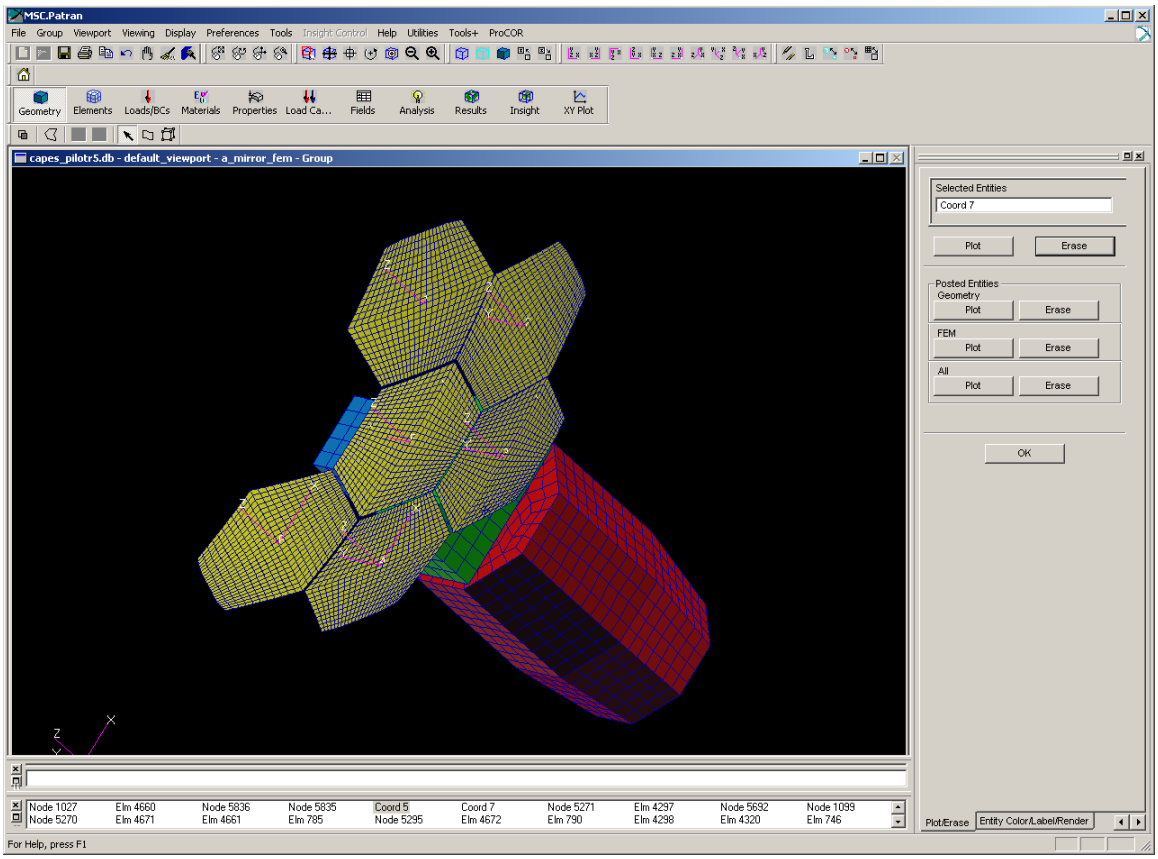
Currently many of the informational fields in the structural model have to be deleted when it is to be used for thermal analysis (depending on the model, this may include material fields, material properties and boundary conditions). However, this deletion usually needs to be done only once, and then the thermal model results can be used to apply a temperature field on the structural model. Hopefully this will be improved in future PATRAN versions. A rough list of what needs to be deleted is as follows, roughly in order (and this may need to be done in the NASTRAN analysis preference):

- structural load cases and BC's
- structural fields (Spatial)
- structural properties and beam section definitions
- structural fields (Material Property)
- structural materials - Composite
- structural materials - 2D, lamina, etc.
- all MPC's

# Structural Interface with Optical Station

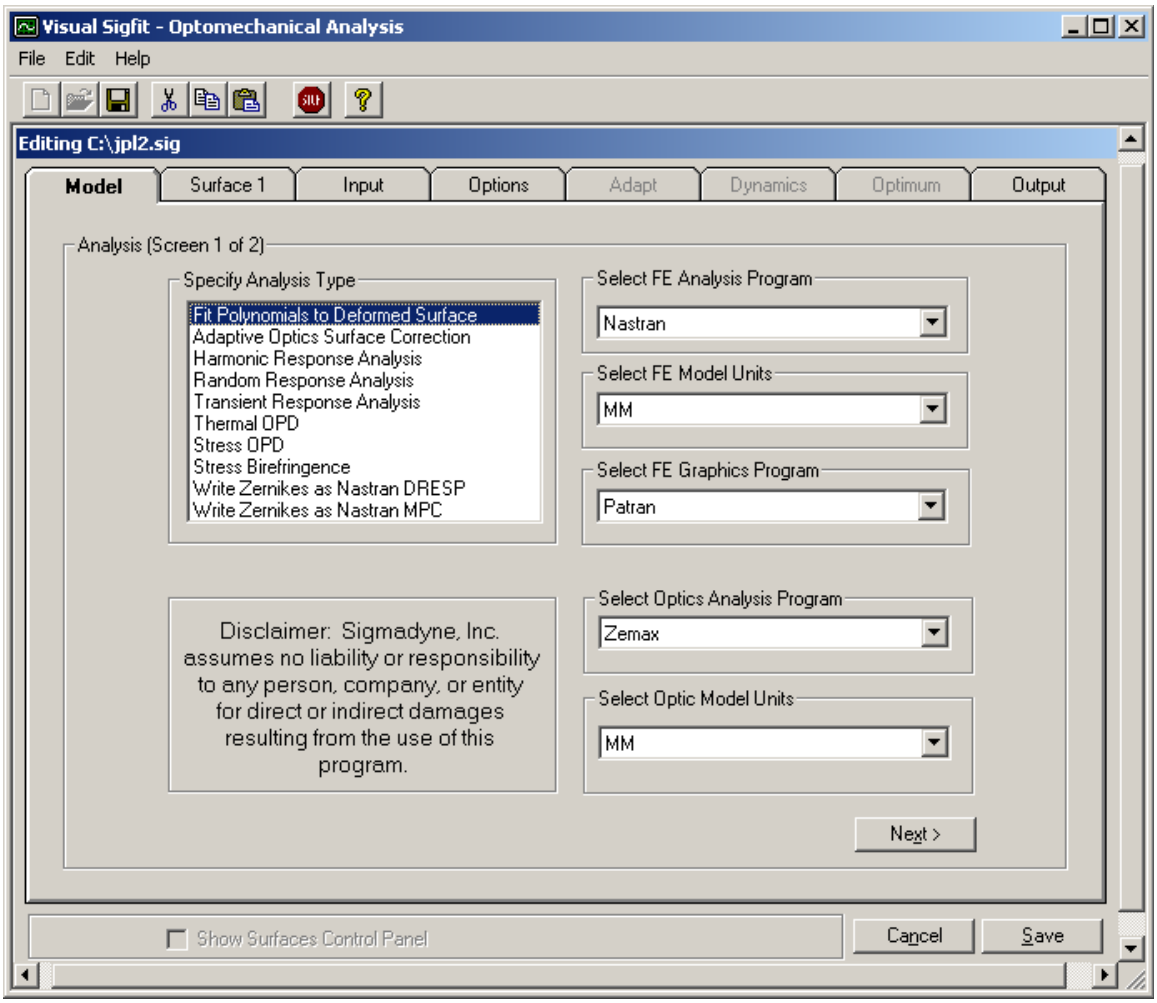
## *Prepare NASTRAN bulk data file for SIGFIT*

The Structural Station should coordinate with the Optical Station to determine coordinate systems and lens prescriptions to be used for the optical analysis.
SIGFIT, a Structural/Optical Interface Code, is used to generate input for ZEMAX from NASTRAN results. SIGFIT requires that there is a rectangular coordinate system located at the center of the mirror vertex. In the CAPES project, a coordinate system was defined at the center of each hex section vertex. This enables the mirror to be analyzed as either a system or as individual segments.

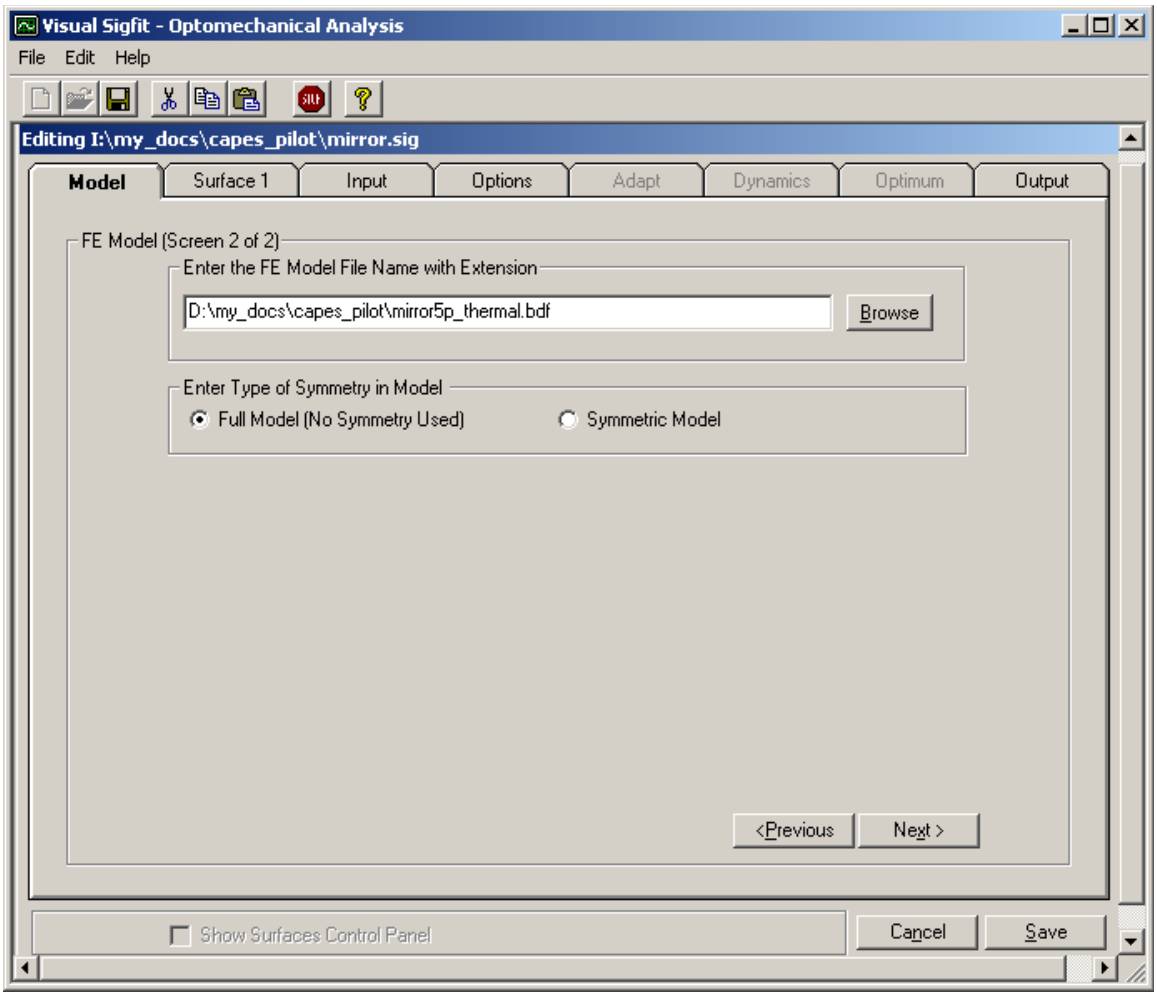



## *Use visual SIGFIT to generate SIGFIT input file*

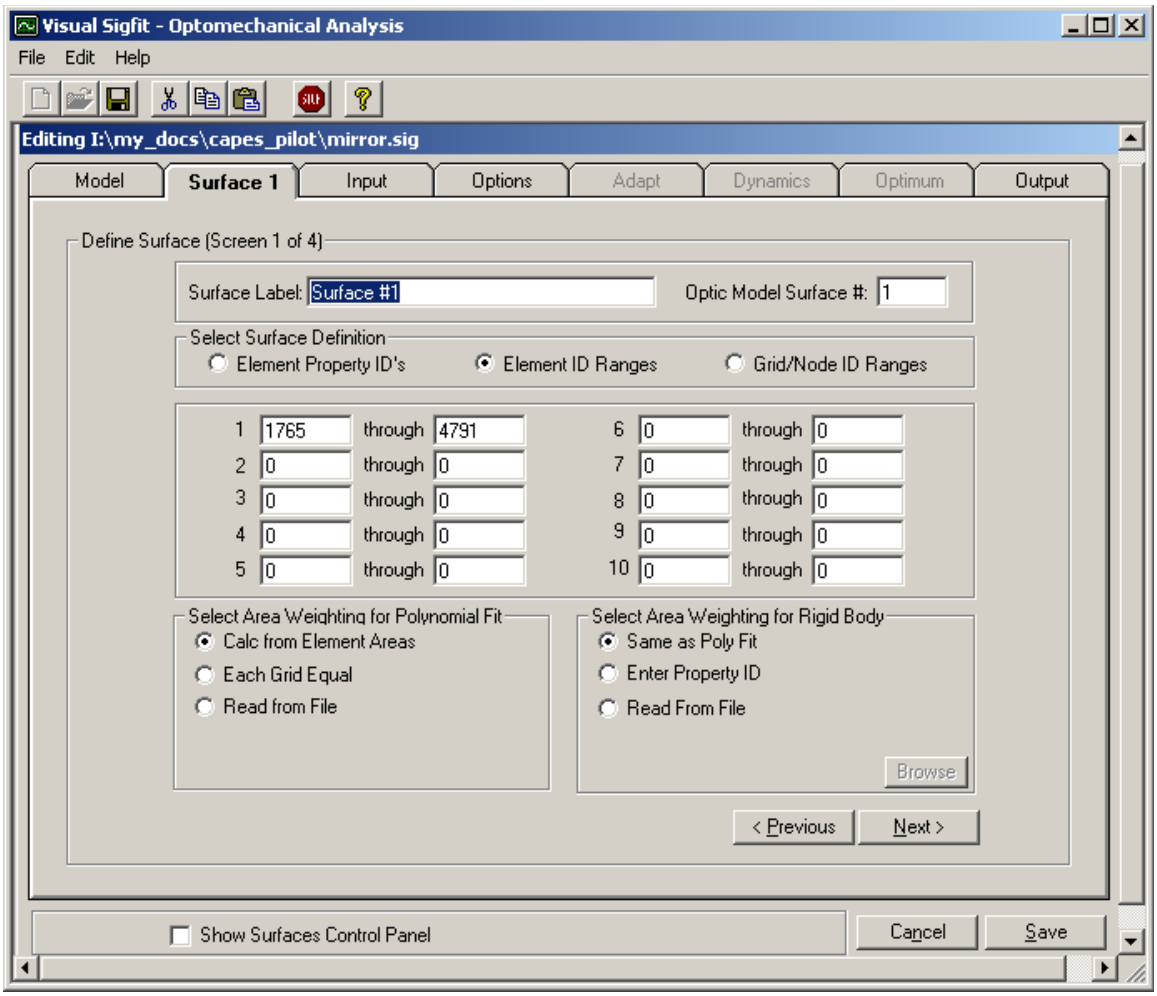
Choose correct units for NASTRAN and ZEMAX on the SIGFIT input form:




SIGFIT reads the NASTRAN bulk data file to get the surface definition. Specify a full model.

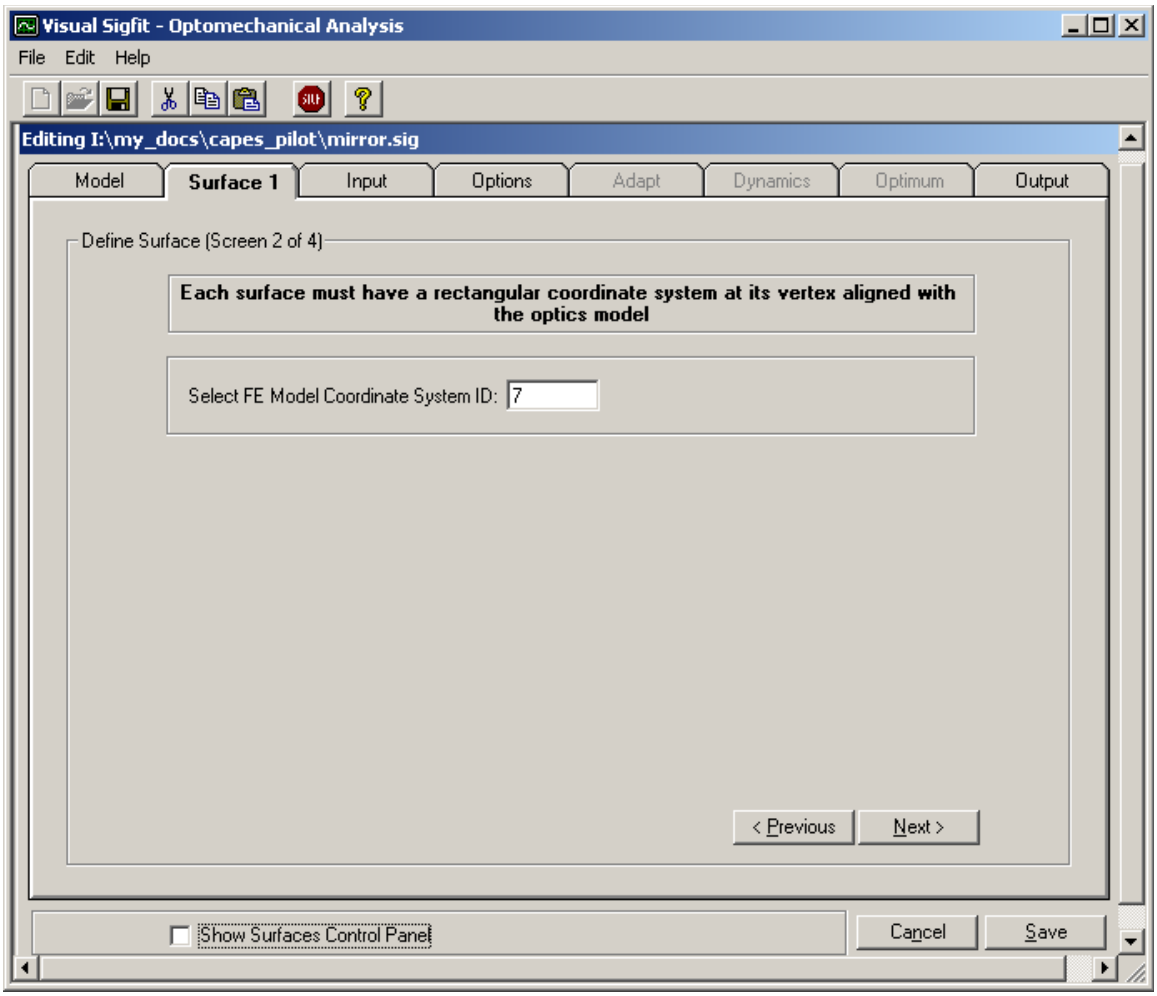



Element Property ID's are used to indicate mirror elements in the NASTRAN bulk data file.

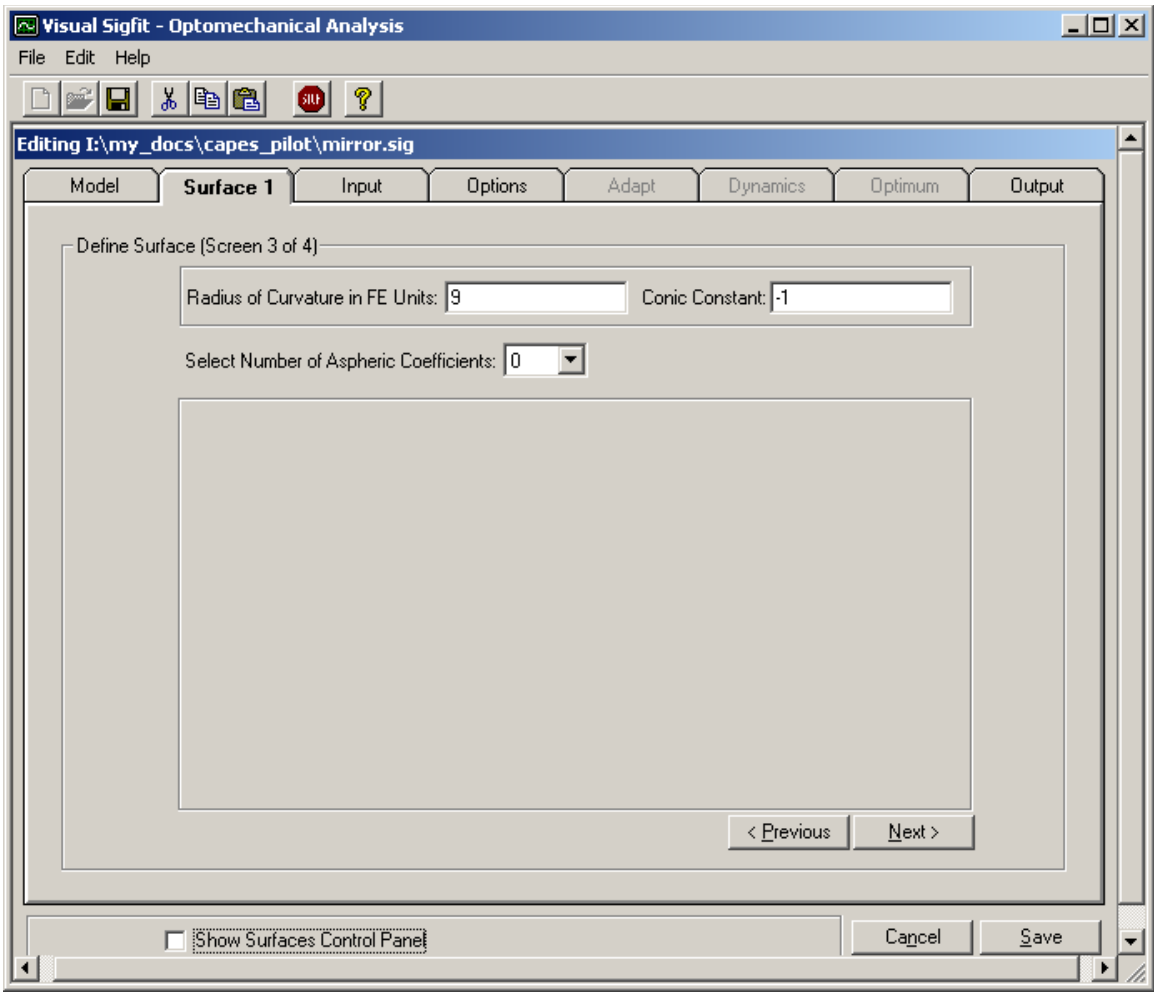



The area weighting is calculated from element areas projected on a flat plane with this selection.

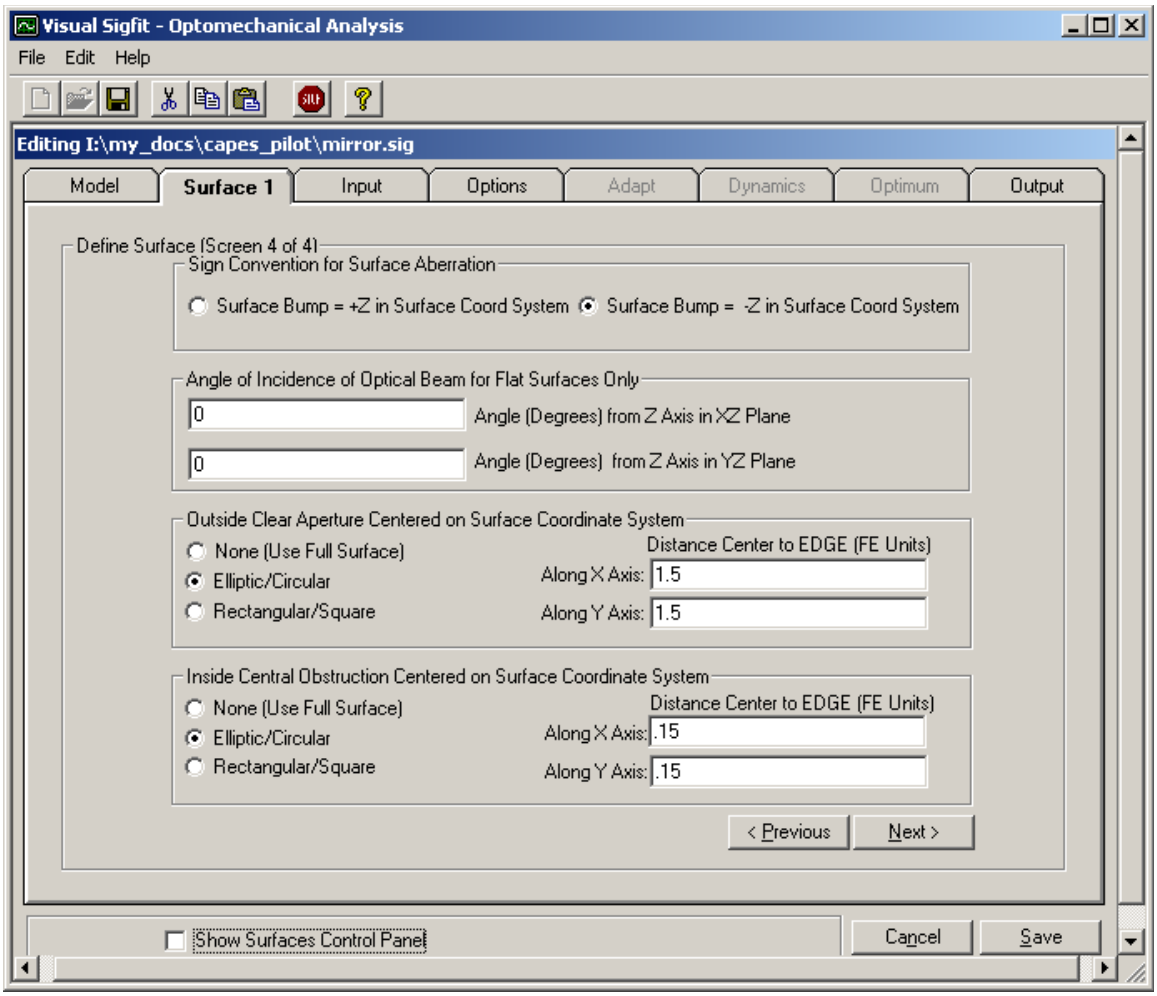
SIGFIT requires that there is a NASTRAN coordinate system located at the center of the mirror vertex. Input the NASTRAN COORD number on this form.



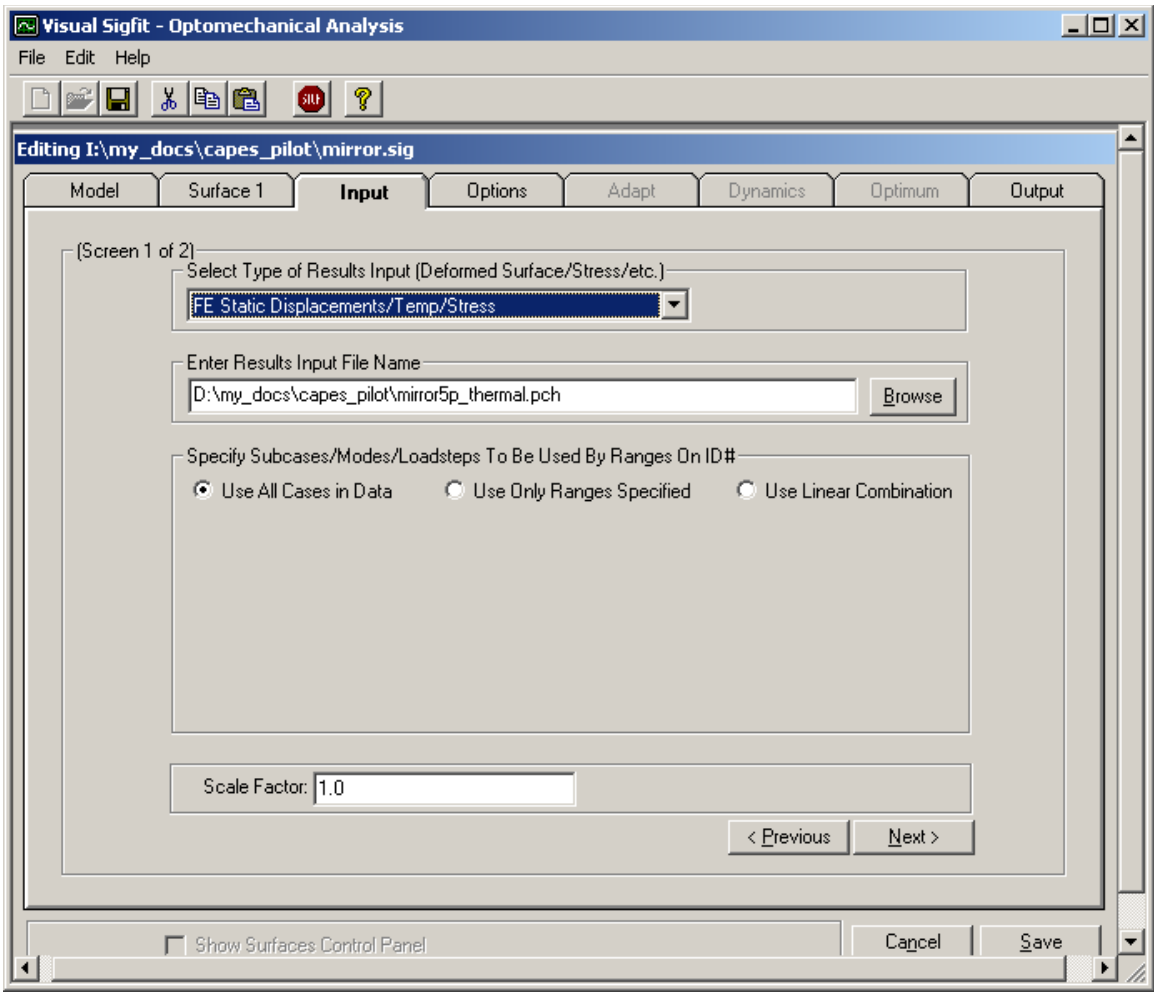

The Optical Station furnishes the mirror prescription, including radius of curvature, conic constant and aperture, which is used as input to SIGFIT.



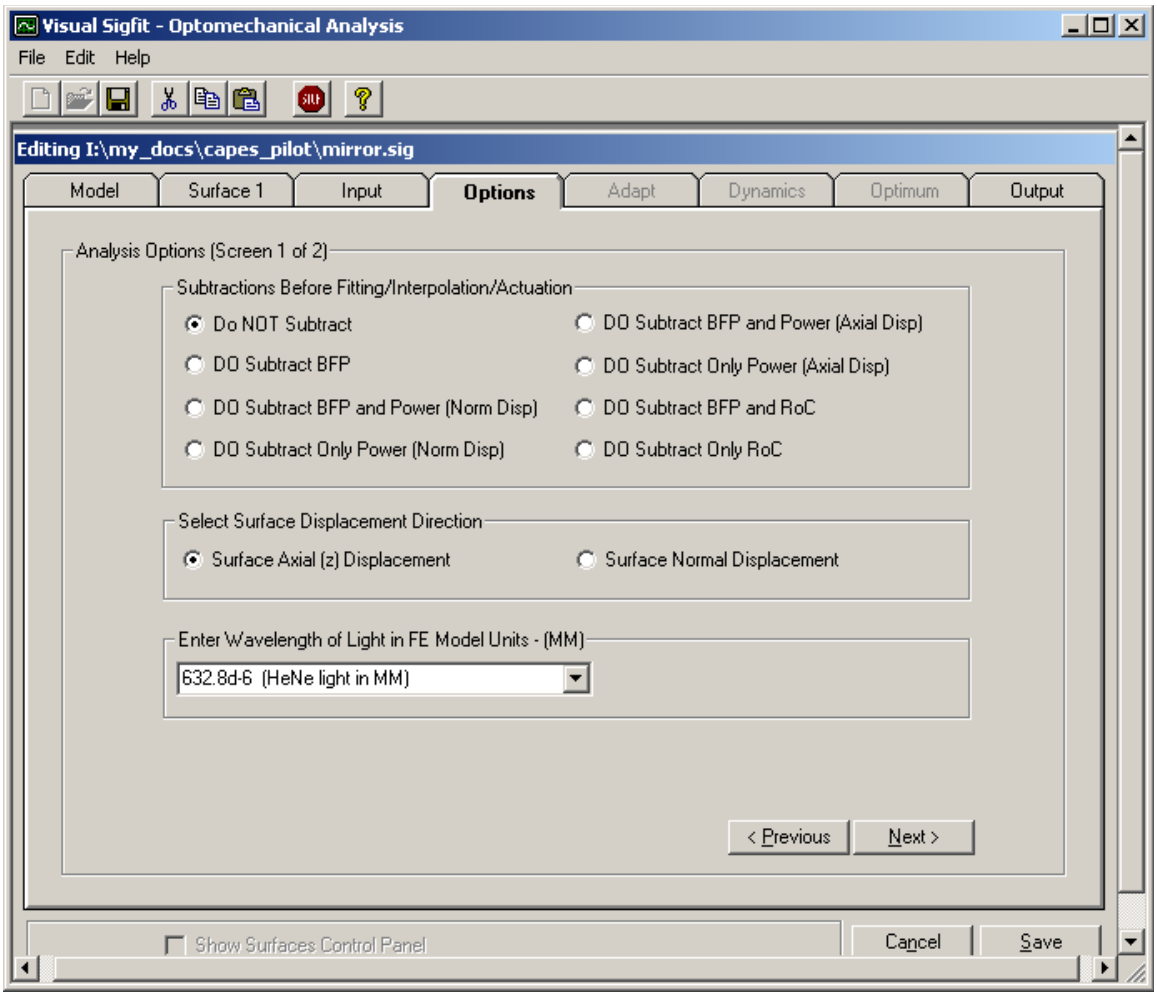

Continue with mirror prescription:



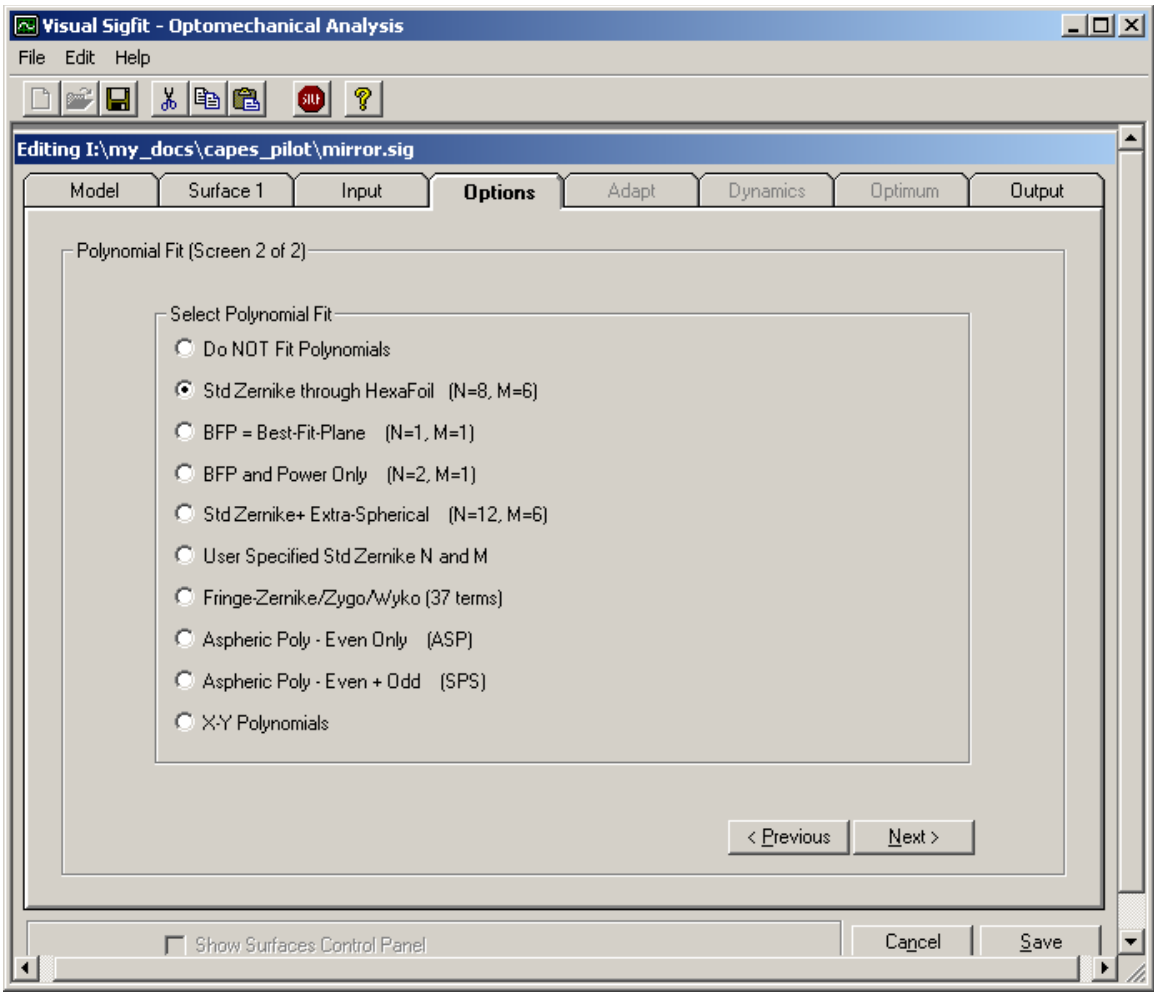

Read NASTRAN results from .pch output file:

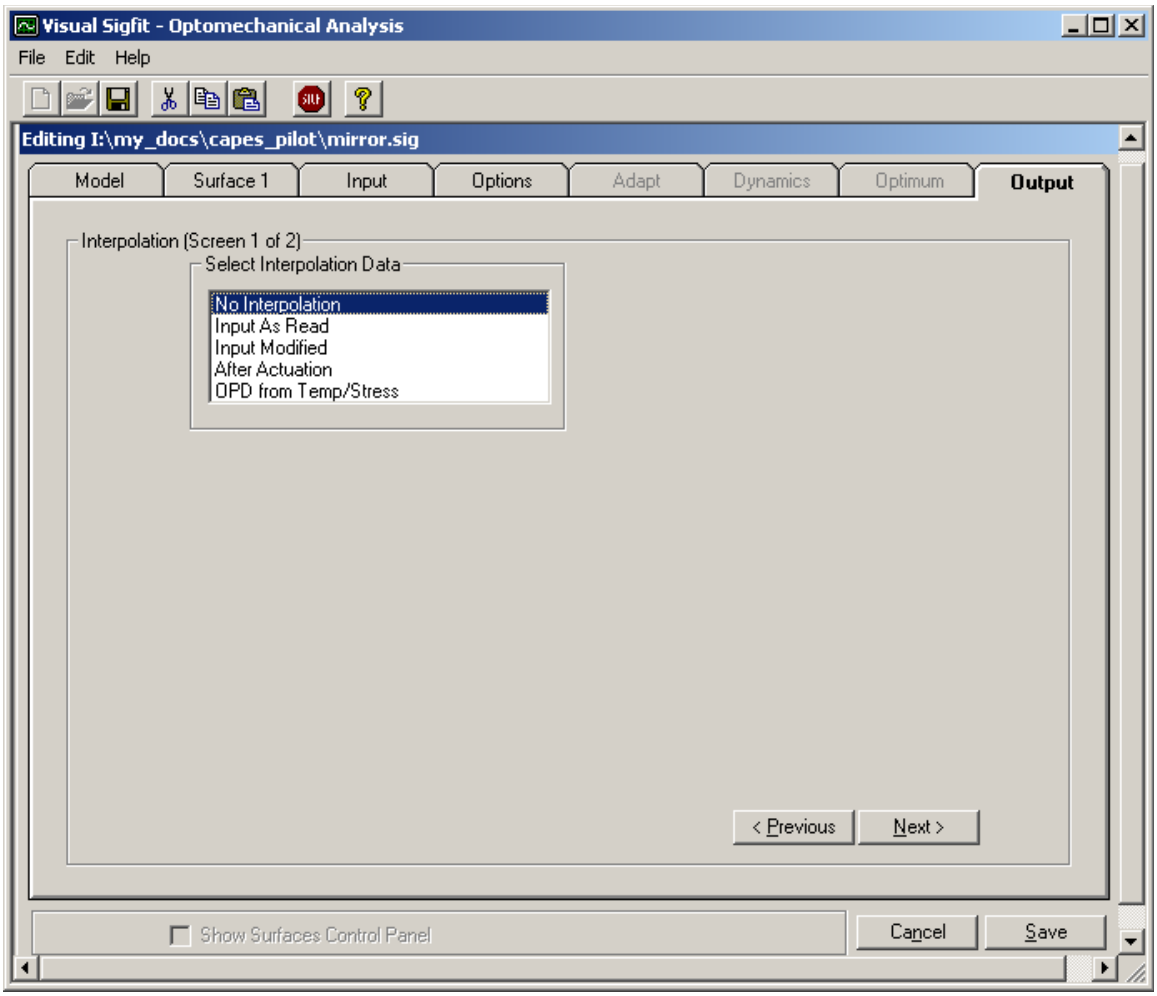



Choose correct options for ZEMAX, i.e. axial sag. CODEV or OSLO have difference specifications. The Optical Station furnishes the correct value for wavelength of light.



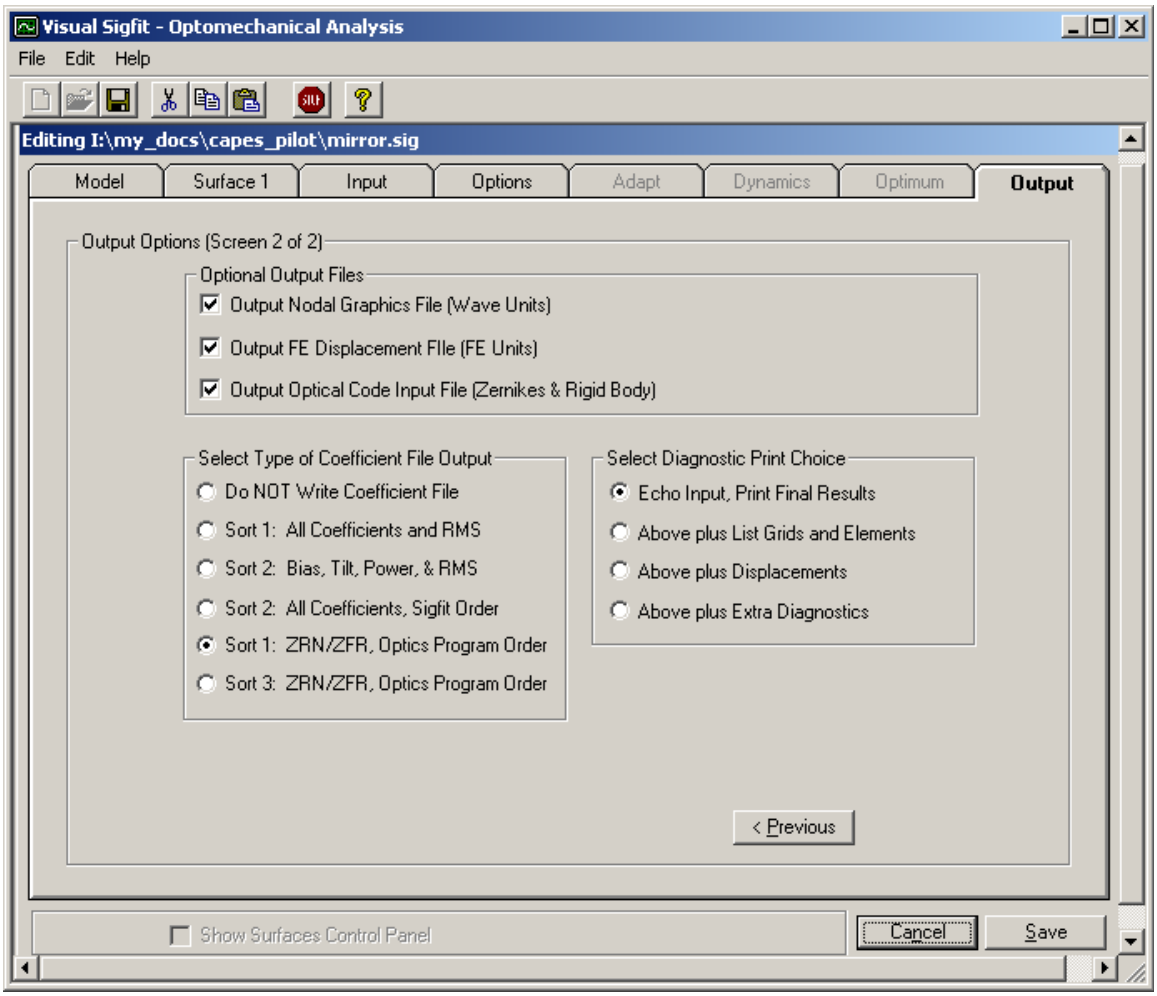

Choose the type of polynomial fit. HexaFoil is not available with the Fringe Zernike, so Std Zernike is chosen for the CAPES Hex mirror.


Visual Sigfit - Optomechanical Analysis
File Edit Help
Editing I:\my_docs\capes_pilot\mirror.sig
Model
Surface 1
Input
Options
Adapt
Dynamics
Optimum
Output
Polynomial Fit (Screen 2 of 2)
Select Polynomial Fit
Do NOT Fit Polynomials
Std Zernike through HexaFoil (N=8, M=6)
BFP = Best-Fit-Plane (N=1, M=1)
BFP and Power Only (N=2, M=1)
Std Zernike+ Extra-Spherical (N=12, M=6)
User Specified Std Zernike N and M
Fringe-Zernike/Zygo/Wyko (37 terms)
Aspheric Poly - Even Only (ASP)
Aspheric Poly - Even + Odd (SPS)
X-Y Polynomials
< Previous
Next >
Show Surfaces Control Panel
Cancel
Save




SIGFIT results can be interpolated back to FEM result for visualization.

Choose outputs:




Save the file, mirror.sig. Batch SIGFIT can now be run to generate the input files for ZEMAX.

-

# Optical Design Process

Cesar Sepulveda

This is a basic outline of the optical design process performed at the Optics Workstation in the IDC. It is in no way a tutorial, since that would require basic knowledge of optical theory, but is intended to give an idea of the design process involved in developing an instrument.

## *Design Guidelines*

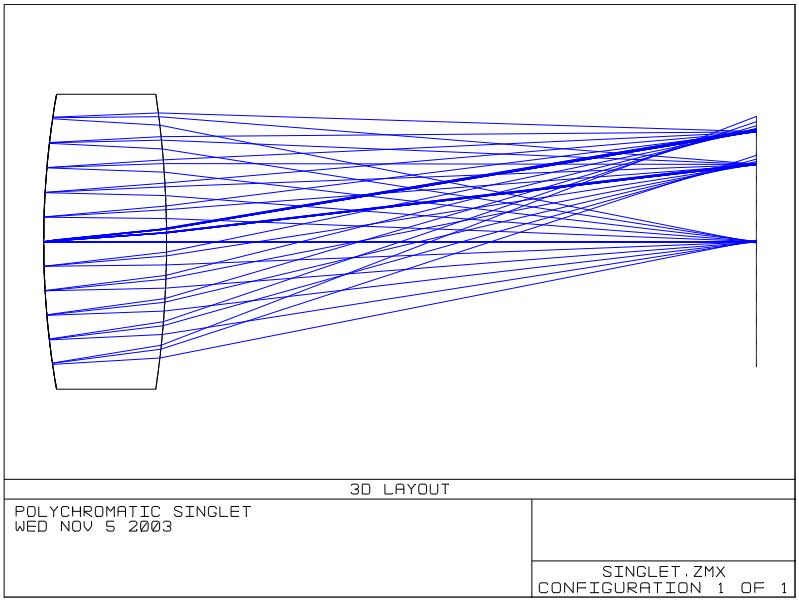
While there are many optical design programs in use at NASA, we will concentrate here on Zemax™ (www.zemax.com) although the approach is similar for CodeV (www.opticalres.com).

Zemax is a powerful program that can be used for design and analysis. It employs the concept of a spreadsheet in which all pertinent information can be dynamically entered and changed.

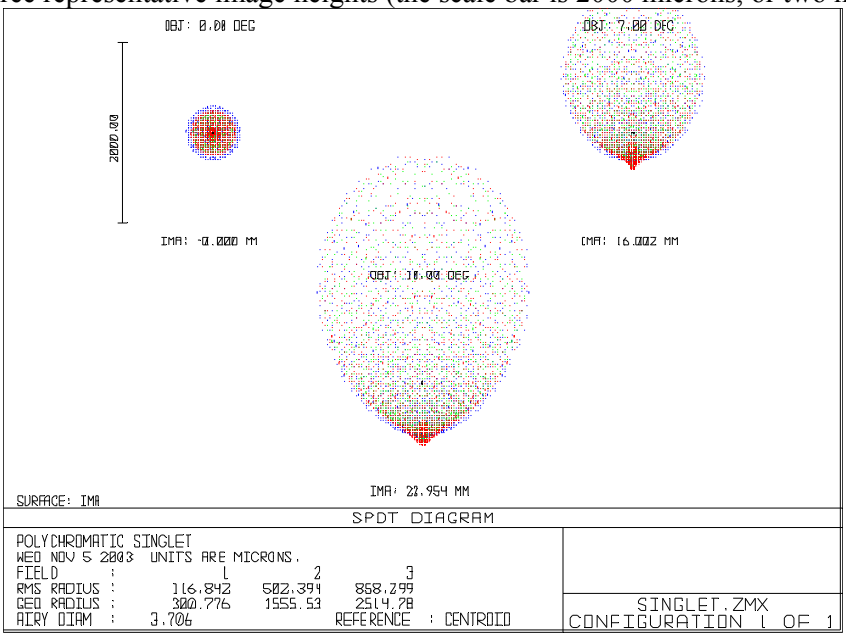
The main quantities in a lens prescription are: radius of curvature, thickness and the type of medium that light propagates through. Any lens design is then a sequence of surfaces containing air, glass, mirrors, etc. As an example, here is a prescription for a singlet lens (some parameters are omitted. In addition, the prescription table shows curvature, that is the inverse of the radius of curvature of the surface, in inverse lens units):

| # | Type | Curvature | Thickness | Glass | |
|---|---|---|---|---|---|
| 0 | STANDARD | 0.0000000E+000 | 1.0000000E+010 | | |
| 1 | STANDARD | 5.7590644E-003 | 2.5000000E+001 | | LAK31 |
| 2 | BINARY_2 | -4.8598673E-003 | 1.2000000E+002 | | |
| 3 | STANDARD | 0.0000000E+000 | 0.0000000E+000 | | |

and here is a picture of that lens :

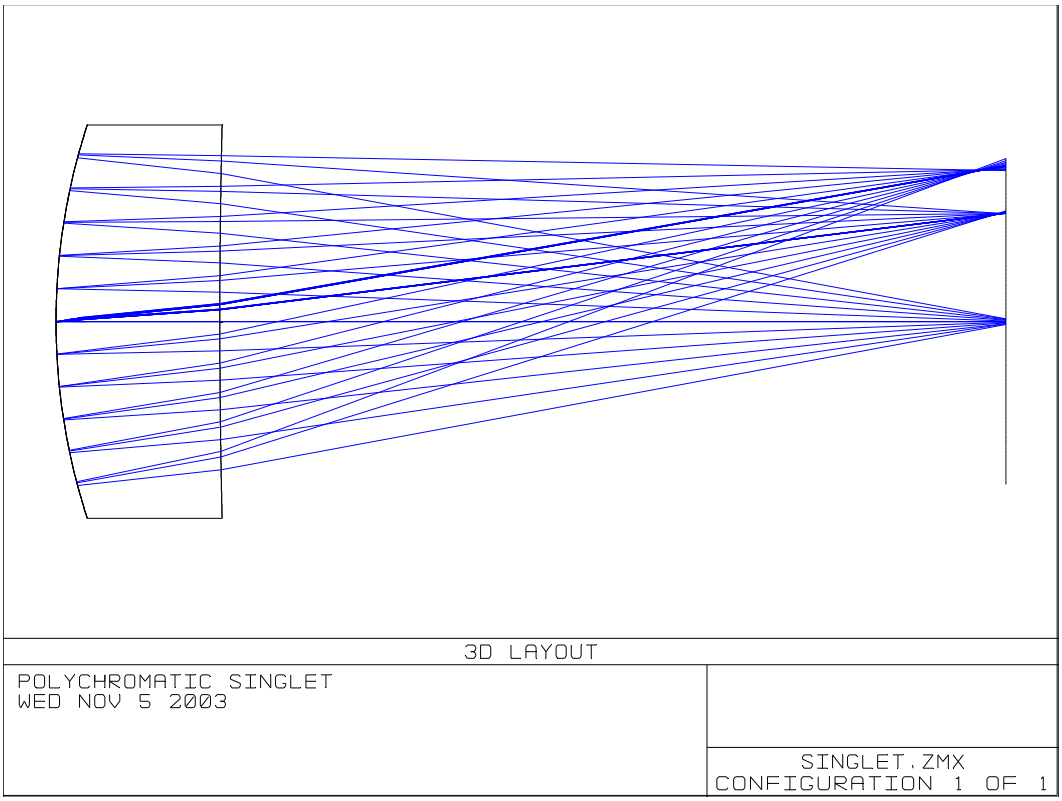


For proper imaging all the rays from a given field angle are supposed to fall on a single point, or close to it. You can see that this is not quite true yet. Here is a spot diagram, that is, a picture of the rays at the focal plane for three representative image heights (the scale bar is 2000 microns, or two mm):

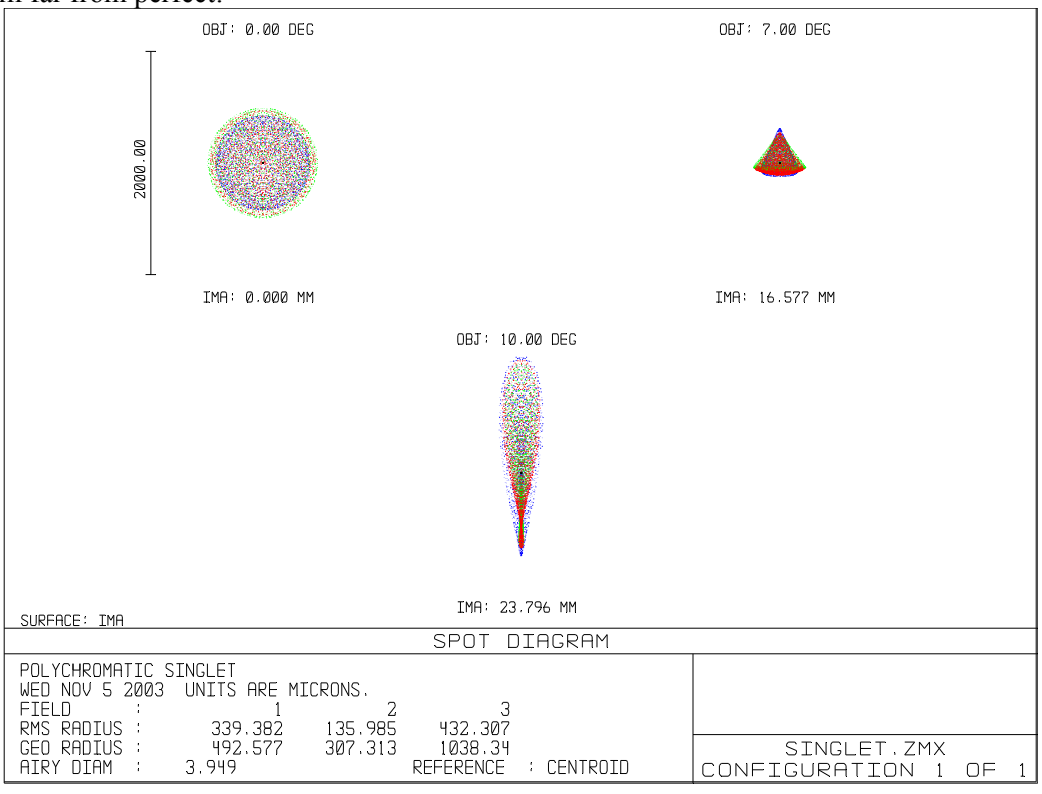



Here is where the optical designer goes to work, improving the quality of the images, that is, making the spot size as small as possible.

One can think of any variables in a lens prescription as degrees of freedom; the more variables, the better one can correct the lens. In this case all we have is the two curvatures and the type of glass used in the lens (lens thickness generally has only a minor effect). Let's see what we can do. In order to improve a lens it is necessary to define a figure of merit, that is, a measure of the degree of optical quality. Zemax has a (very complex!) merit function editor, which fortunately for the designer, does a lot of the heavy work automatically. In this case we employ a "ray-trace" merit function, in which we try to bring all the rays to focus as tightly as possible for each of the points in the image. This is the 'spot size' criterion, in which the sum of the squares of the ray distances to the common spot centroid is minimized, usually with additional constraints like keeping the focal length of the system constant. Let's try this. The two lens curvatures will be used as variables, and the focal length constrained to equal 125 mm. If the focal length is not controlled, it generally will grow without limit while Zemax tries to minimize the spot size!

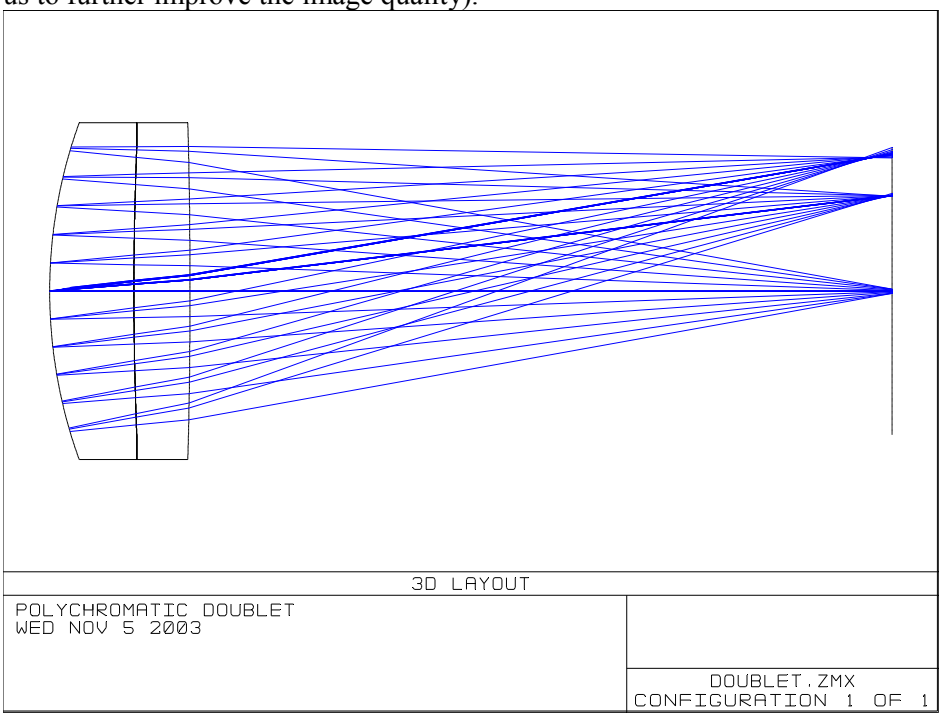


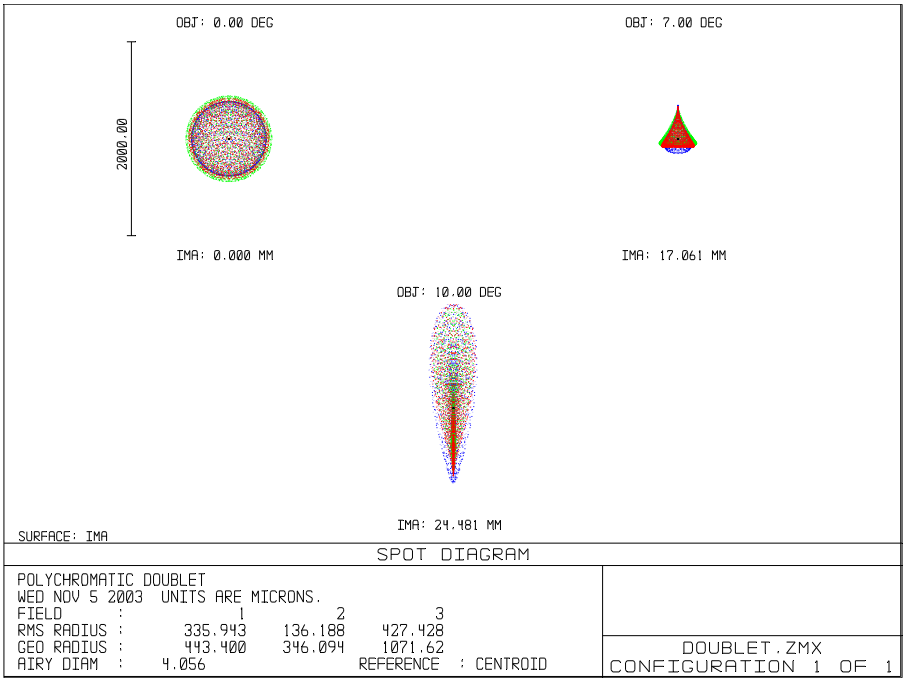

We allowed curvatures to change, while maintaining the overall focal length (the distance from the lens to the image plane) constant. Although the rays now focus at a smaller spot on the focal plane, the lens is still far from perfect:



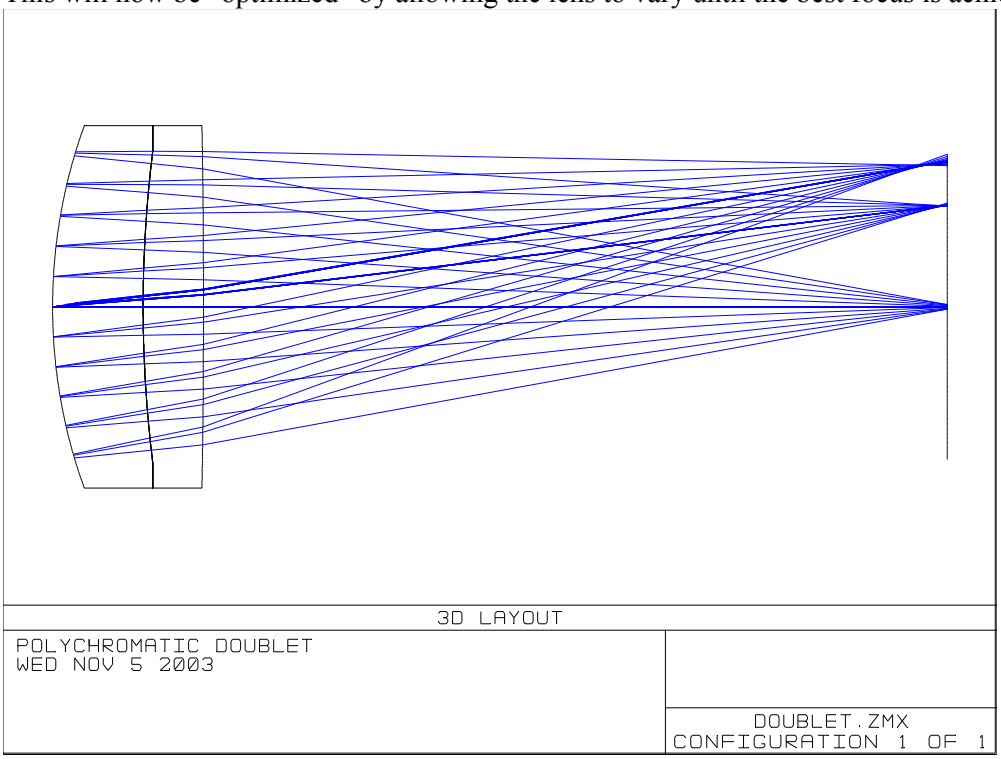
The main reason is that glass has a different refractive index (or "bending power") for each wavelength, so generally more glass types are required to compensate for that. We have run out of degrees of freedom here, so more are needed.

Now we will try a cemented doublet lens (two glass elements, three independent surfaces). We will see that the image quality can now be made much better, since we now have more variables, or degrees of freedom (two glasses, plus three curvatures). We will start with an arbitrary doublet, and improve on that. (Sometimes it is necessary to separate the lens elements, which gives us one more variable, and enables us to further improve the image quality).

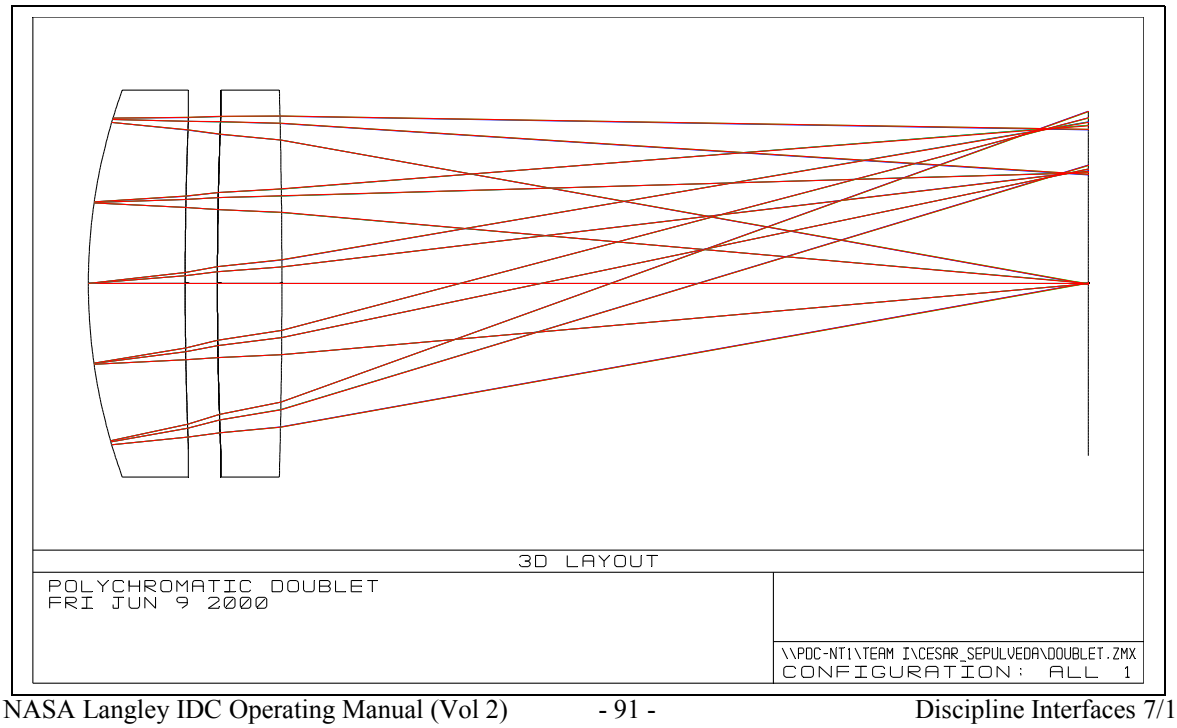


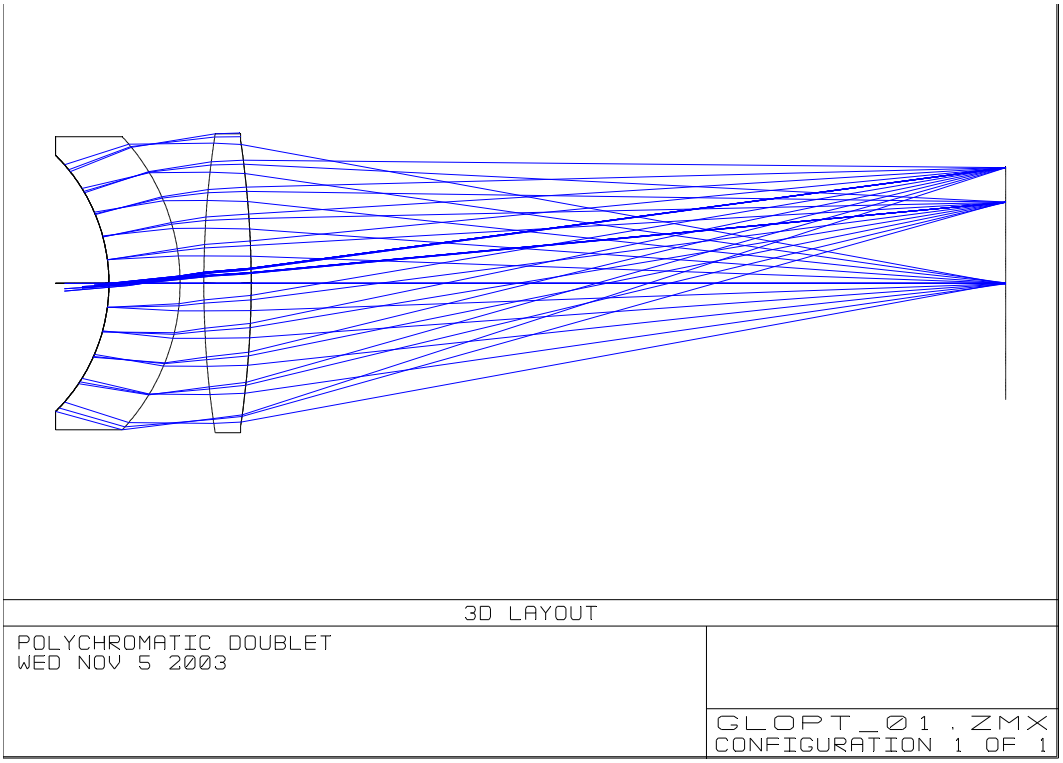
This will now be “optimized” by allowing the lens to vary until the best focus is achieved:



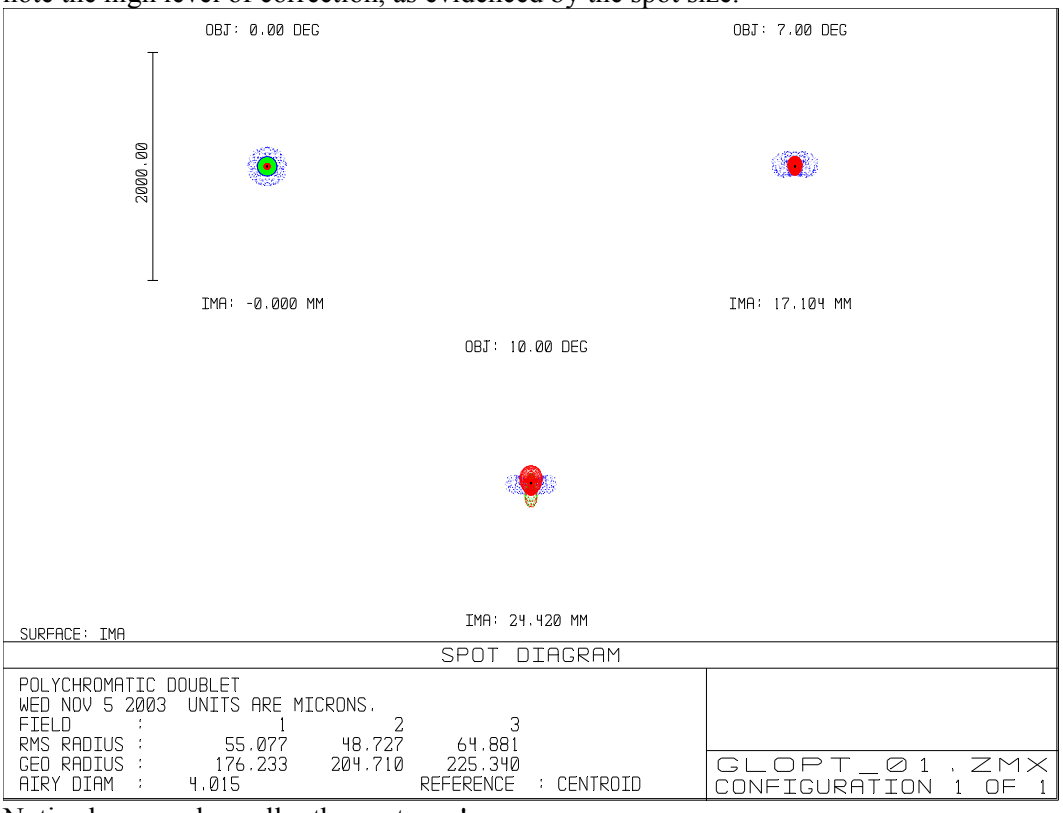

Although the glasses were allowed to vary, as well as the curvatures, there is not much improvement. This means that for this application a doublet is simply not good enough. This happens even to the best designers! One tries to use the simplest design that will do the job, then go from there. So let us now “break” the lens apart and try one more variable (four curvatures, two glass types, one airspace). Here is the starting point. We will see where this takes us…



and this is the design we get in the end. The front element has a slightly extreme shape, but

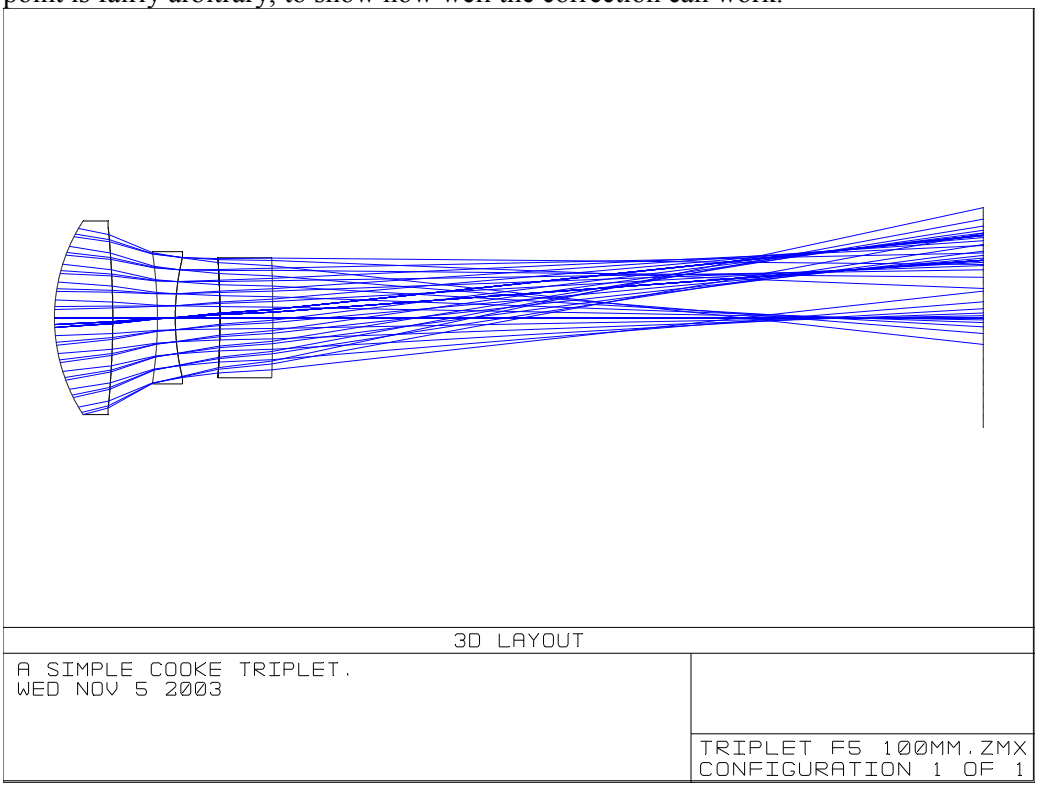



note the high level of correction, as evidenced by the spot size:

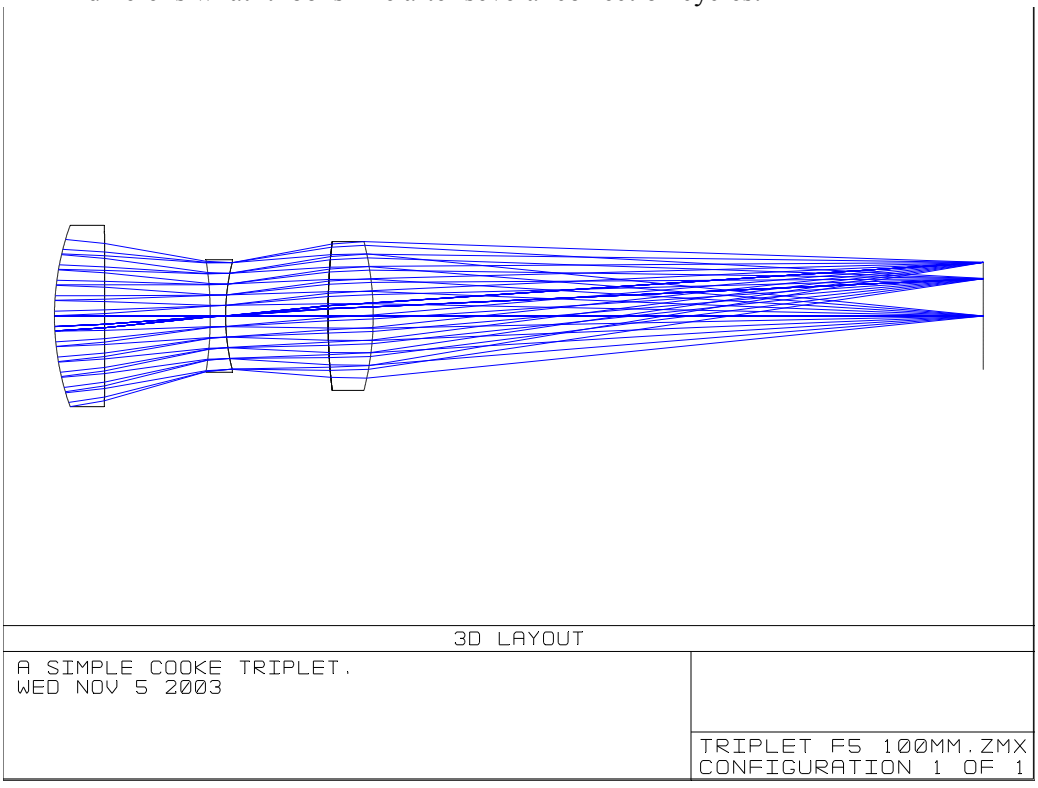


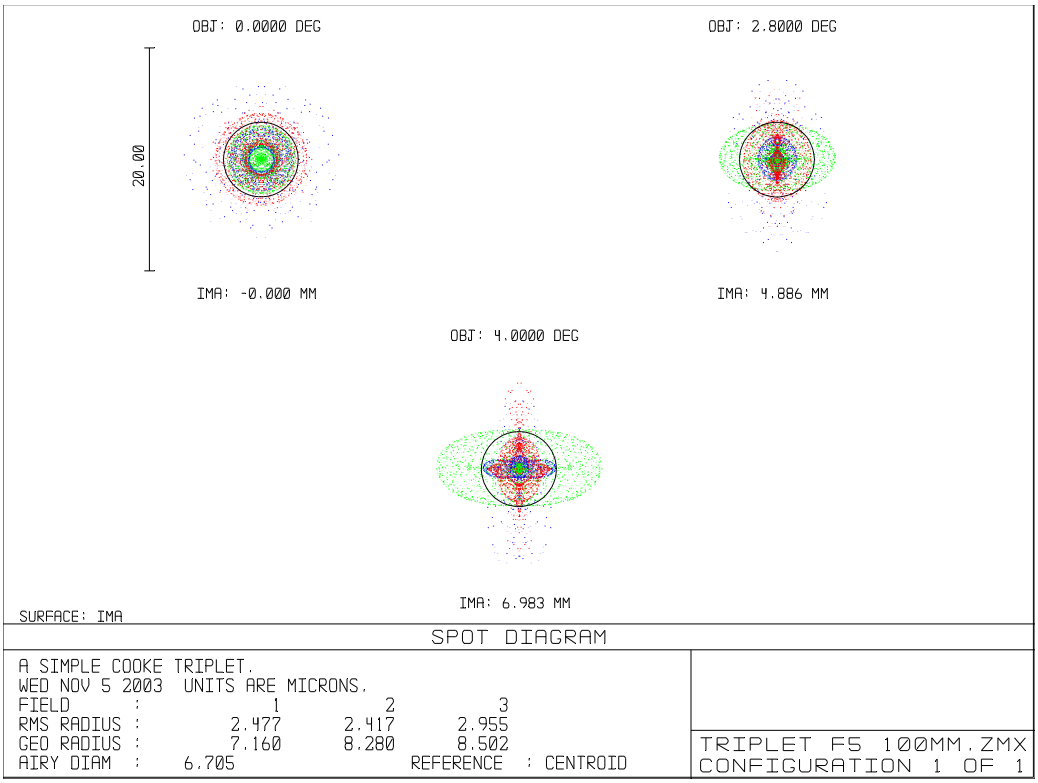

Notice how much smaller the spots are!

We will end this demonstration with a triplet lens (three glasses, six surfaces, two airspaces). The starting point is fairly arbitrary, to show how well the correction can work.




And here is what it looks like after several correction cycles:






The spot size is now about 10 microns, or about a thousand times better that was possible with a singlet or even an airspaced doublet. The circles in the spot diagram represent the size of the Airy disk, which is the smallest theoretical spot size for a given wavelength and imaging conditions. This means that we can do no better than that.

## *Conclusion*

There are many types of optical instruments, each with its own basic requirements. Here you have seen just a hint of what is possible in lens design using powerful software to optimize lenses. Since a full tutorial would be outside the scope of this demonstration, the user is referred to the Zemax demo and tutorial at http://www.zemax.com/demo_z.html

Although the software makes the design process easier, brute force is no substitute for knowledge; it is still necessary to know what to do, that is, which starting point to choose, which variables to use, and especially when to stop! We hope this gives at least a basic idea of what is involved in designing an optical instrument.

## *Optics Checklist*

**These lists are not necessarily comprehensive, but should provide a minimum of the input and output parameters necessary for optical system development**

RECEIVABLES:
Class of instrument: Telescope, Camera, Spectrometer
Field of view: Narrow, Medium, Wide,
Type of field: Point, Line, Square, Rectangular
Field of View (FOV) dimensions: Angular, Linear
Instantaneous FOV (iFOV): Angular, Linear
Type of instrument: Refractive, Reflective, Catadioptric (both)
Spectral range: Visible, Infrared, Ultraviolet)
Radiometric Sensitivity, Throughput.
Construction (if known) : Deployable, Monolithic, Lightweighted
Parameters: System aperture, Focal length, F/number
Particular requirements: Accesible intermediate images or pupils, Cooled optics, Cryogenic detectors)
Environmental: High or low temperature operation, Underwater or high-pressure atmosphere, Corrosive or high-radiation environment.
Quality of optical surfaces: Superpolished, Diamond-turned, Replicated
Pointing requirements: Stability, accuracy, repeatability

DELIVERABLES:

As a minimum: A completed optical design showing manufacturing tolerances, sensitivity to variations in manufacturing and assembly parameters: tip/tilt, decenter, despace, surface irregularity.

Supporting CAD documents. Generally the optical design is carried over to the Mechanical CAD Workstation as a combination of an IGES line work file exported directly from within Zemax plus an ACIS (SAT) file generated by means of the TracePro software (www.lambdares.com). This is done by reading the native .ZMX format file into TracePro, then exporting it as a .SAT file. It is very useful to have an 'Optics CAD' folder to hold these files so Optics and Mechanical have a common location for convenient access.

If a multi-configuration file is to be translated, say a beamsplitter with two optical paths) it is necessary to convert it to individual single-configuration files ("Channel 1", "Channel 2", etc) since TracePro only "sees" the main configuration and not the alternates.

Supporting documentation showing the decision process used to arrive at the particular design.

Discussions regarding the suitability of the chosen design. Can it be built, or does it require a separate technology development effort? Can it be tested?

Proposed assembly and integration procedure (this always works as a sanity check!)

Proposed test and validation procedures (same as above)

Powerpoint or similar file(s) can be very useful for capturing and documenting the details of the design process

# Electronics Design to Mechanical CAD model

These are notes on importing an ORCAD electronics board layout into Pro/Engineer (Pro/ECAD), and then into PATRAN for analysis. Additions and suggestions for this page are welcome.

## *Import into Pro/Engineer from OrCAD & Pro/ECAD*

1. fix the file names; they arrive from OrCAD as file.brd & file.lib
   - rename file.brd to file.emn (contains board layout info, 2nd line says Board)
   - rename file.lib to file.emp (contains component info, 2nd line says Library)
2. grab ECAD license:
   - misc, float option, ECAD, done
3. import unpopulated circuit board as a Pro/E part:
   - part, import (name it what you like), ecad in, idf 2.0, board, file.emn
4. make the board the first component of an assembly
   - aa (mapkey for EDB start_assembly, which has a coord system; or you can create a new assembly & give it a coord system), component, assemble bare board (assembly by coordinate system works very well)
5. then import components:
   - interface, import, ECAD, IDF 2.0, default options, select assy coord system
   - when Pro/E asks for Neutral file name: file.emn
   - when Pro/E asks for Profiles file name: file.emp
   - import process takes off

*potential problems (Robert Dillman, 757-864-7177, r.a.dillman@larc):*

1. tested w/ OrCAD-output IDF files; one has .brd extension, other .lib; if don't rename the file extensions, you need to type the full file name when Pro/E asks (file.brd, instead of ? and picking file)
2. Pro/E won't read input from files w/ capital letters in name; rename first or you'll get error "cannot open file"
3. Pro/ECAD reads in each component, gives it a default coord system named ECAD_DEFAULT, and asks which coord system to use to assemble the component; you can either click "ECAD_DEFAULT" once per component, or add "ECAD_COMP_CSYS_DEF_NAME ecad_default" to your config.pro (tells it to use the default coord system; I tried, but couldn't get it to change what name it uses)
4. Pro/ECAD doesn't like components used in library (.lib) file to have spaces in their names; if it yacks while reading file, look at the file, and if spaces are the problem get them renamed in OrCAD or use unix sed.
5. sometimes when trying to read components, Pro/E creates a file saying how to fix import problems, called ecad_hint.add; rename this file to ecad_hint.map, and re-run the import, some things (but not all) will work better.

## *Thermal analysis of electronics board in PATRAN*

- When done correctly, the parts of an assembly will come in with their group names defined by the part file names from Pro/E.
- In general, the board will come in as a trimmed solid. Unless you specifically need the holes, it will be easier to handle as a (green) native PATRAN surface. Just select the edges of one side of the solid to define it as a surface. Also, at least in the one example I ran, the board came in with an incorrect thickness, so you may want to check the thickness before you use it for the plate property. This is another reason to go with the plate instead of solid formulation. If you need the holes, just define one side of the solid as a trimmed surface (then you will only be able to get a triangular mesh).
- **WARNING:** to save yourself time, you may not want to not post the group(s) that contain the board solid, or you might even have the designer suppress some layers in the Pro part before you pick it up, because for some reason the trimmed solid board will generally slow the display in PATRAN down immensely (it also makes the import take longer -- can be several hours).
- All chips will come in as trimmed solids, so you will only be able to get a tet mesh (with the current version of PATRAN). If you have to have a solid brick mesh, you could select the bottom surfaces of all chips at once and do an extrude -- or select tops and bottoms to do a 2-Surface Solid Create.
- Don't use the properties of glass-filled epoxy from PATRAN -- the k is too low for most boards -- and the properties seem to be such that you can get instability in trying to solve with high-power chips.
- The simplest and most accurate way to attach the chips to the board is to use contact convection. Since you won't want any of the nodes to be equivalenced together, the best way is to translate the chips off the board by some distance (greater than your tolerance) before you mesh them. Then apply properties and mesh. You can apply contact conduction between the geometries by picking the chip bases as region 1 (easy to do in a side view), and the board as region 2. The h value you select will define the chip temperatures, so make sure you include the effect of epoxies, pin connections, air convection, etc.
- Layers in the board will have a large impact since the amount of copper in each region will vary for each layer. You don't have to do separate layers to be accurate, just make sure that each region of the board has a Cp and k appropriate for the percentage of copper present.
- Heat sinks and bolts will have huge effects on chip temperatures and are sometimes not included in the Pro/ECAD file. Make sure you find out about any extras of this type.

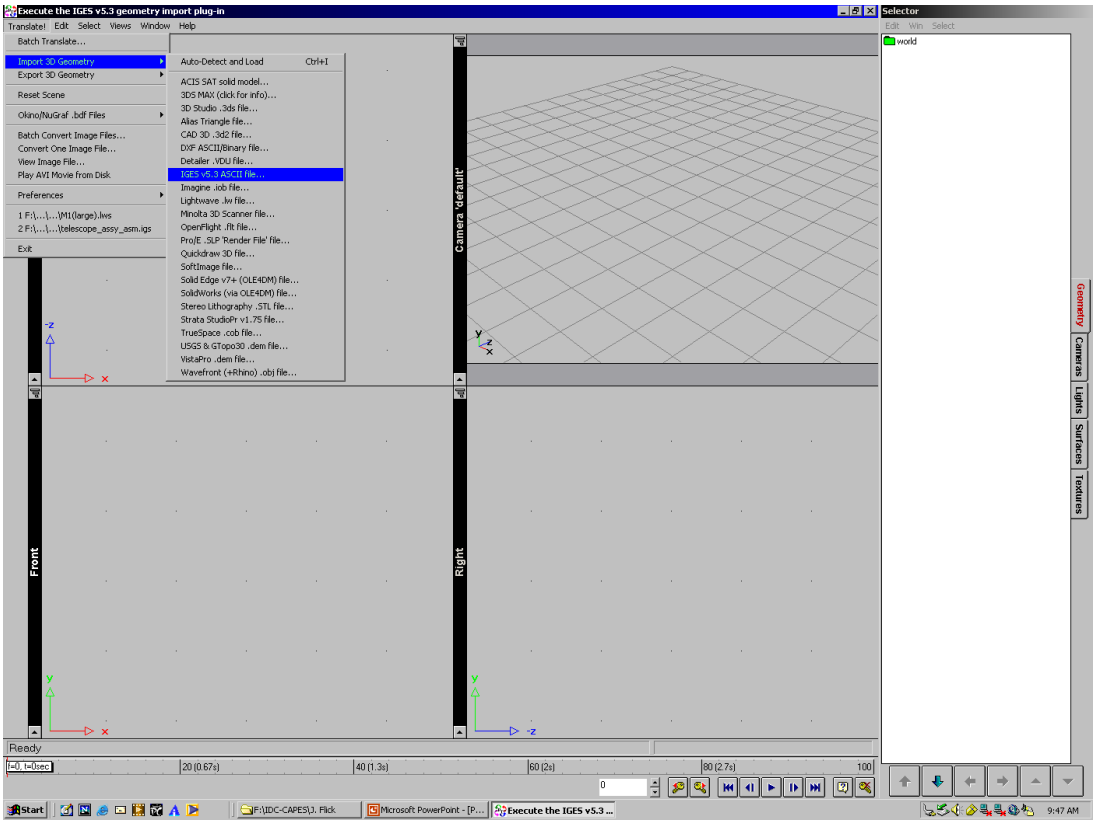
*An example board in Pro/E and PATRAN:*

## Mechanical CAD model to Orbital/Environment Station

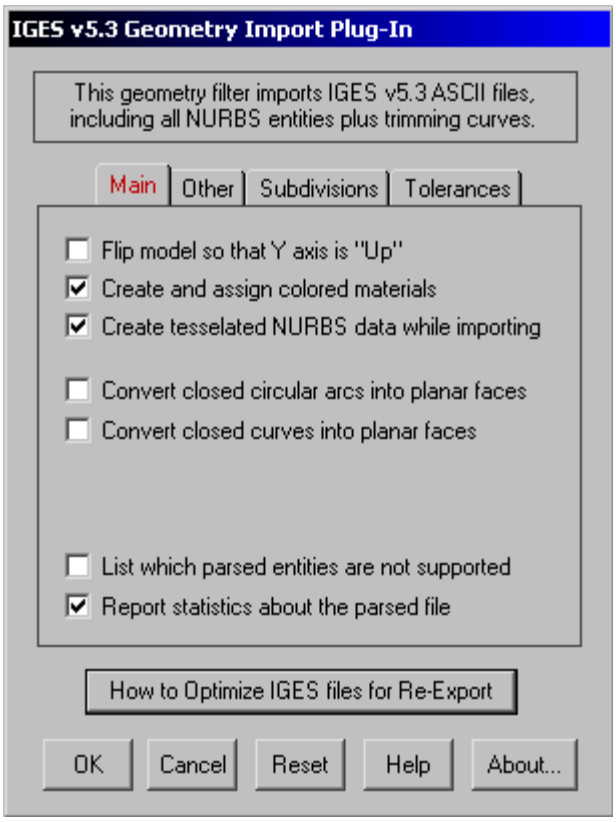
***Step 1:*** *Obtain exported ProE model file (preferred file is IGES v5.3 ASCII file), through shared drive.*

***Step 2:*** *Using PolyTrans to convert the IGES file into Lightwave (.lwo file)*

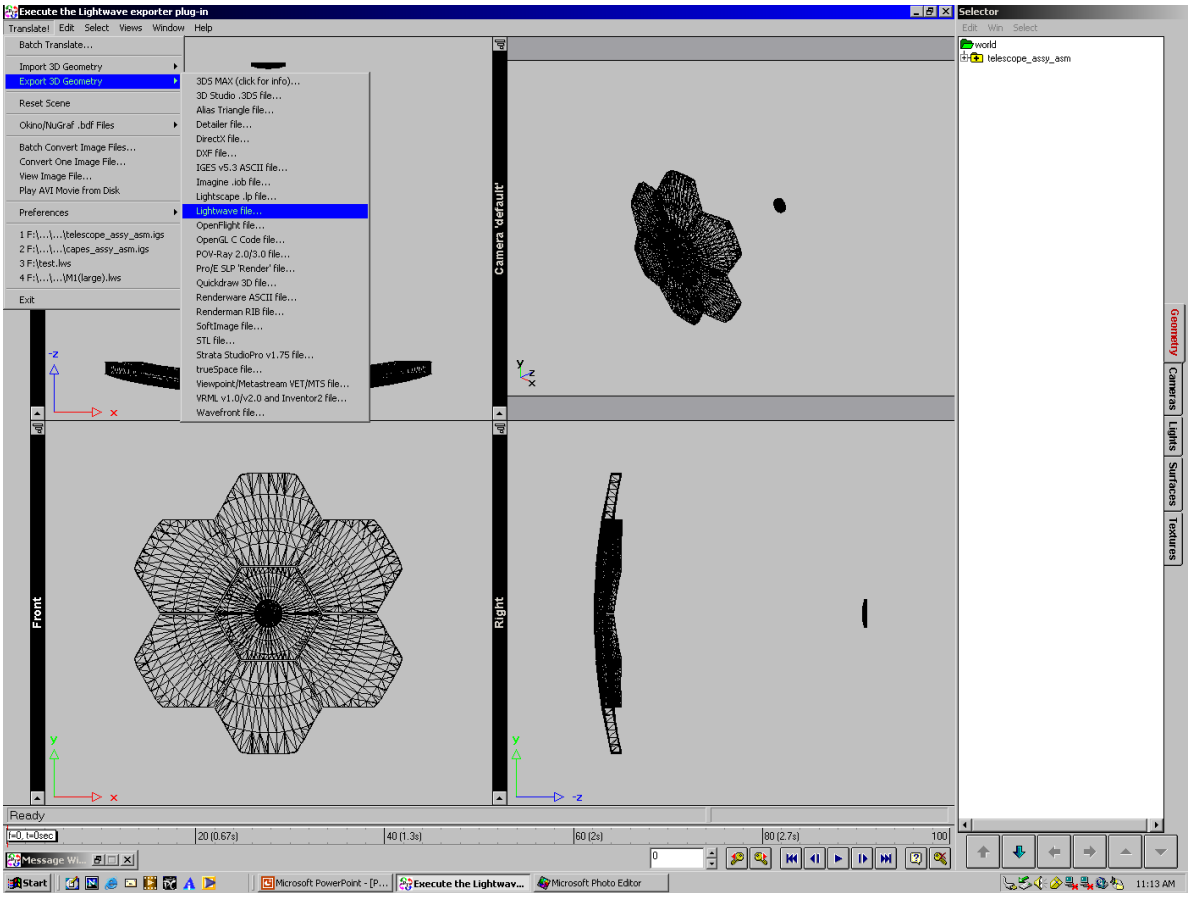
- From the "Translate!" drop down menu highlight "Import 3D Geometry". See figure below.
- Select the file format to import.
- This file format selected is the CAD geometry (eg. IGES .igs file) you want to eventually import into STK.


Execute the IGES v5.3 geometry import plug-in
Translate!
Edit
Select
Views
Window
Help
Batch Translate...
Import 3D Geometry
Export 3D Geometry
Reset Scene
Okino/NuGraf .bdf Files
Batch Convert Image Files...
Convert One Image File...
View Image File...
Play AVI Movie from Disk
Preferences
1 F:\...\...\M1(large).lws
2 F:\...\...\telescope_assy_asm.igs
Exit
Auto-Detect and Load Ctrl+I
ACIS SAT solid model...
3DS MAX (click for info)...
3D Studio .3ds file...
Alias Triangle file...
CAD 3D .3d2 file...
DXF ASCII/Binary file...
Detailer .VDU file...
IGES v5.3 ASCII file...
Imagine .iob file...
Lightwave .lw file...
Minolta 3D Scanner file...
OpenFlight .flt file...
Pro/E .SLP 'Render File' file...
Quickdraw 3D file...
SoftImage file...
Solid Edge v7+ (OLE4DM) file...
SolidWorks (via OLE4DM) file...
Stereo Lithography .STL file...
Strata StudioPr v1.75 file...
TrueSpace .cob file...
USGS & GTopo30 .dem file...
VistaPro .dem file...
Wavefront (+Rhino) .obj file...
Camera 'default'
Front
Right
Selector
Edit
Win
Select
world
Geometry
Cameras
Lights
Surfaces
Textures
Ready
20 (0.67s)
40 (1.3s)
60 (2s)
80 (2.7s)
100
Start
Execute the IGES v5.3 ...
9:47 AM

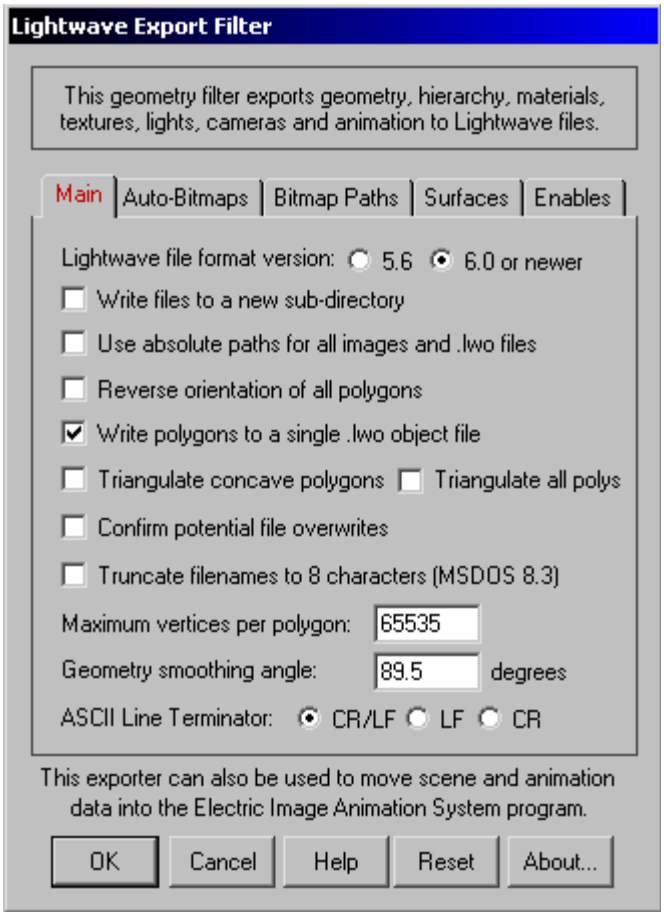
- After selecting the file to import into PolyTrans the dialog box shown here will appear.
- Make sure all appropriate boxes are checked. The boxes selected from the Main tab shown in the figure can be used as defaults by the user. Generally, the options on the other tabs can be left at their defaults.
- Select OK when satisfied.




- Once you have successfully imported the CAD geometry (eg. IGES .igs file), it can be exported into another file format.
- Select “Export 3D Geometry” from the “Translate!” drop down menu. See figure at right.
- For STK we will need to convert to a Lightwave file.

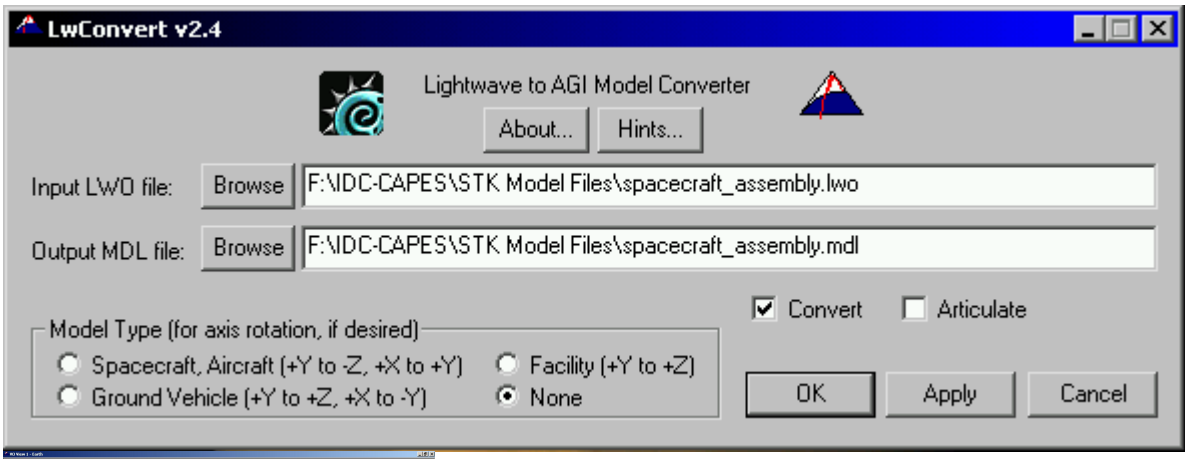

Execute the Lightwave exporter plug-in
Translate!
Edit
Select
Views
Window
Help
Batch Translate...
Import 3D Geometry
Export 3D Geometry
Reset Scene
Okino/NuGraf .bdf Files
Batch Convert Image Files...
Convert One Image File...
View Image File...
Play AVI Movie from Disk
Preferences
1 F:\...\...\telescope_assy_asm.igs
2 F:\...\...\capes_assy_asm.igs
3 F:\test.lws
4 F:\...\...\M1(large).lws
Exit
3DS MAX (click for info)...
3D Studio .3DS file...
Alias Triangle file...
Detailer file...
DirectX file...
DXF file...
IGES v5.3 ASCII file...
Imagine .iob file...
Lightscape .lp file...
Lightwave file...
OpenFlight file...
OpenGL C Code file...
POV-Ray 2.0/3.0 file...
Pro/E SLP 'Render' file...
Quickdraw 3D file...
Renderware ASCII file...
Renderman RIB file...
SoftImage file...
STL file...
Strata StudioPro v1.75 file...
trueSpace file...
Viewpoint/Metastream VET/MTS file...
VRML v1.0/v2.0 and Inventor2 file...
Wavefront file...
Camera 'default'
Front
Right
Selector
Edit
Win
Select
world
telescope_assy_asm
Geometry
Cameras
Lights
Surfaces
Textures
Ready
t=0 t=0sec
20 (0.67s)
40 (1.3s)
60 (2s)
80 (2.7s)
100
Message Wi...
Start
Microsoft PowerPoint - [P...
Execute the Lightwav...
Microsoft Photo Editor
11:13 AM

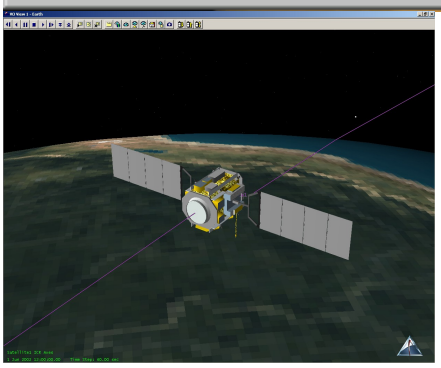
- After selecting the Lightwave file format to export to, the dialog box shown here will appear.
- Make sure the "Write polygons to a single .lwo object file" box is selected and click OK.
- PolyTrans will then convert the file into two separate Lightwave files (.lws & .lwo). The .lwo file is of importance for STK. The files can be saved in any convenient directory.



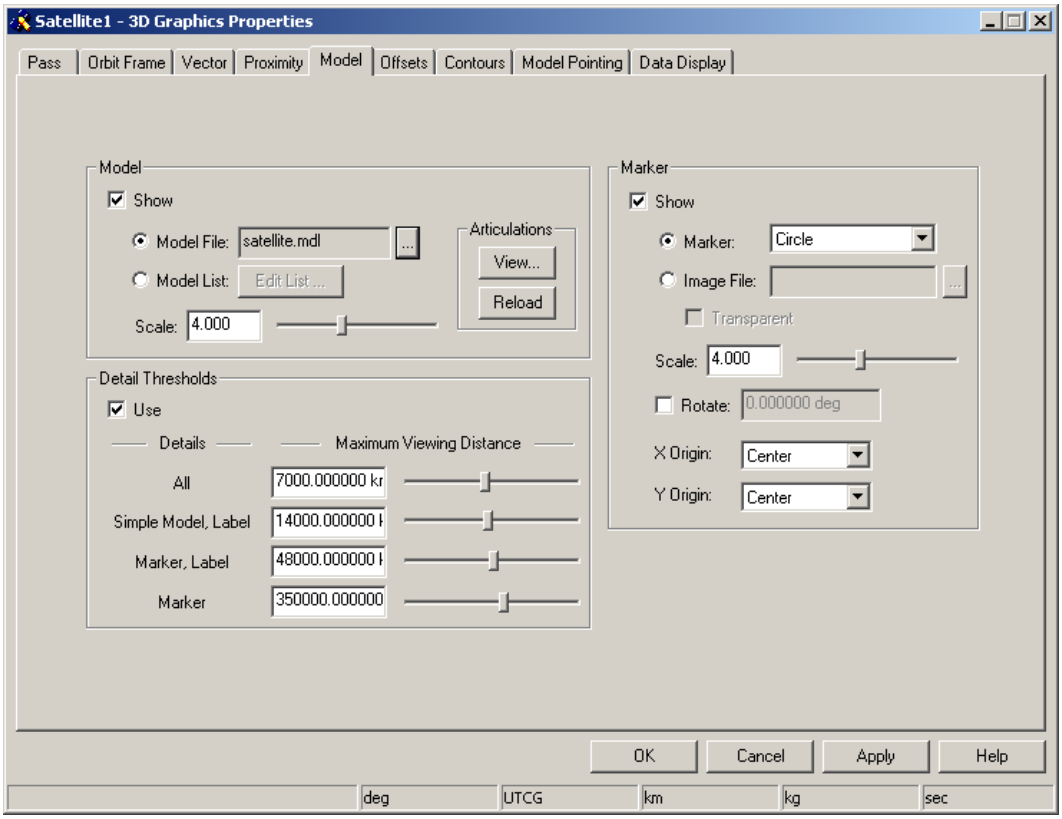

***Step 3:*** *Using STK's file converter, convert the IGES file into a model file (.mdl file*

- Open the Lightwave to AGI Model Converter (LwConvert.exe on the desktop of the Orbital/Environmental Station machine). See figure at right.
- Browse to find the PolyTrans converted Lightwave .lwo file of the CAD geometry. Make sure the "Model Type" option is set to None.
- A file of the same name with the .mdl extension will appear in the "Output MDL file" area. Click OK to create a .mdl format file of the CAD geometry.
- This .mdl file can then be imported into STK through the 3D Graphics menu for the scenario spacecraft.

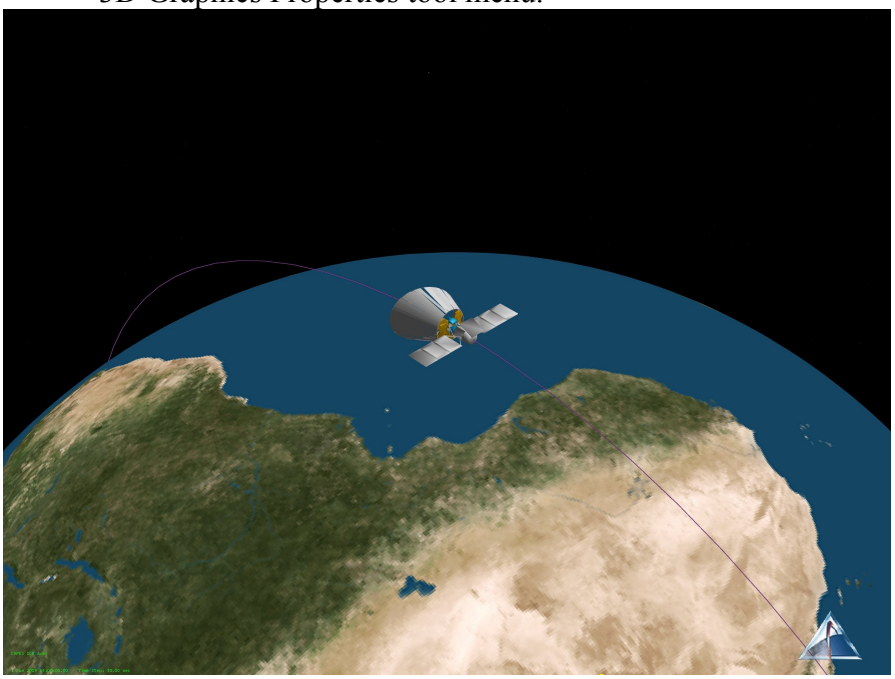


- When a satellite is created in STK a default geometry is assigned to it. See figure at right.
- Once the user has converted their spacecraft geometry file into the .mdl format it can be exported into STK, replacing the default satellite geometry.




- Pictured is an example of a different .mdl file that was created from an IGES file following the procedure outlined earlier in this document.
- may need to adjust the scale of the satellite. This can be done under the Model tab of the 3D Graphics Properties tool menu.

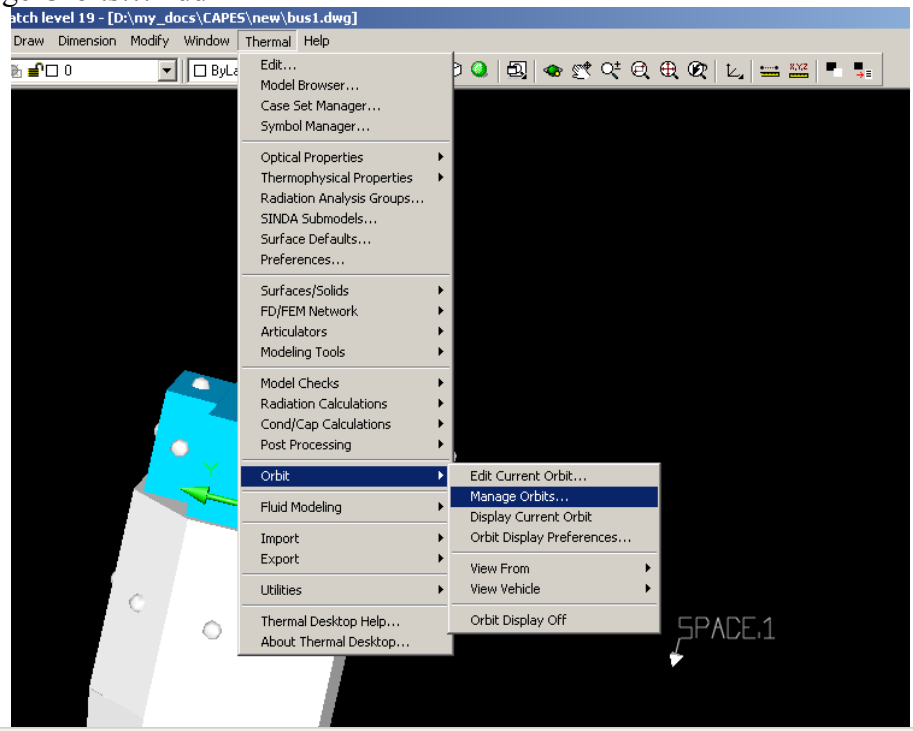


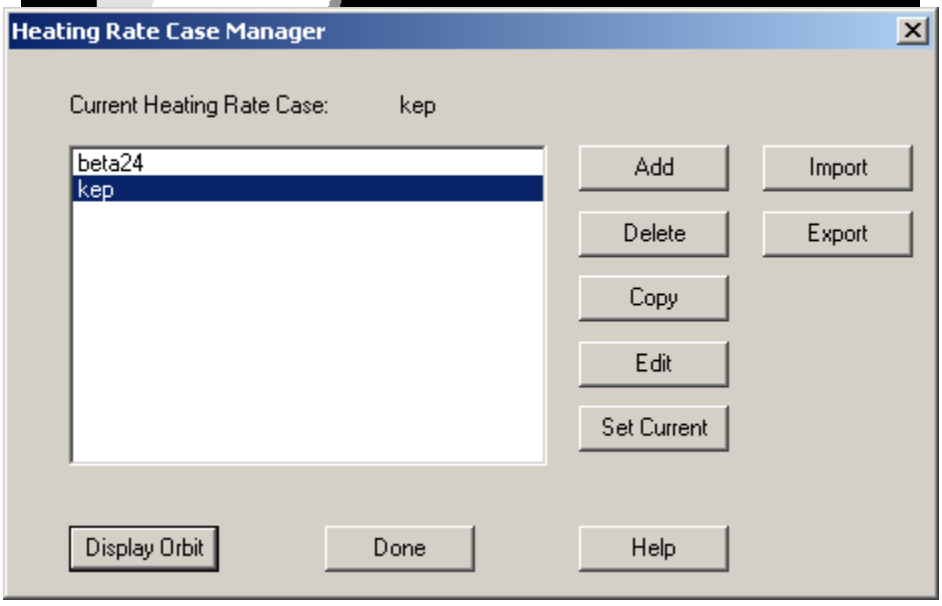
# Orbital/Environment Info to Thermal Station

Information transfer is via text/Excel file, and values are re-entered in Thermal Desktop

**Screen shots in Thermal Desktop/RadCAD:**

Thermal…Manage Orbits…Add



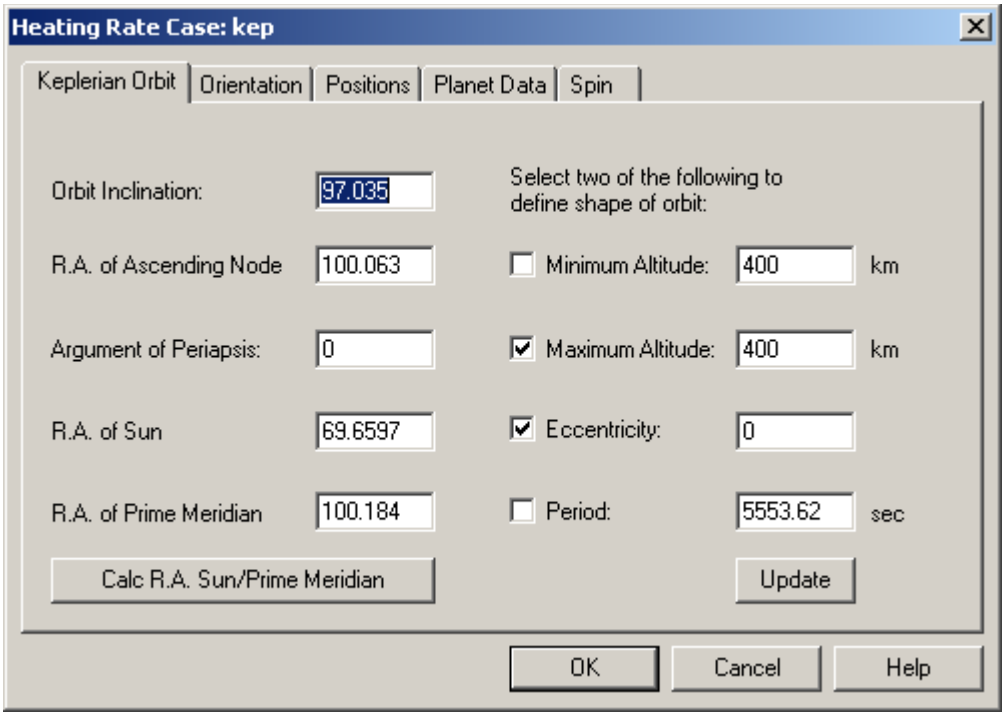




Select Keplerian

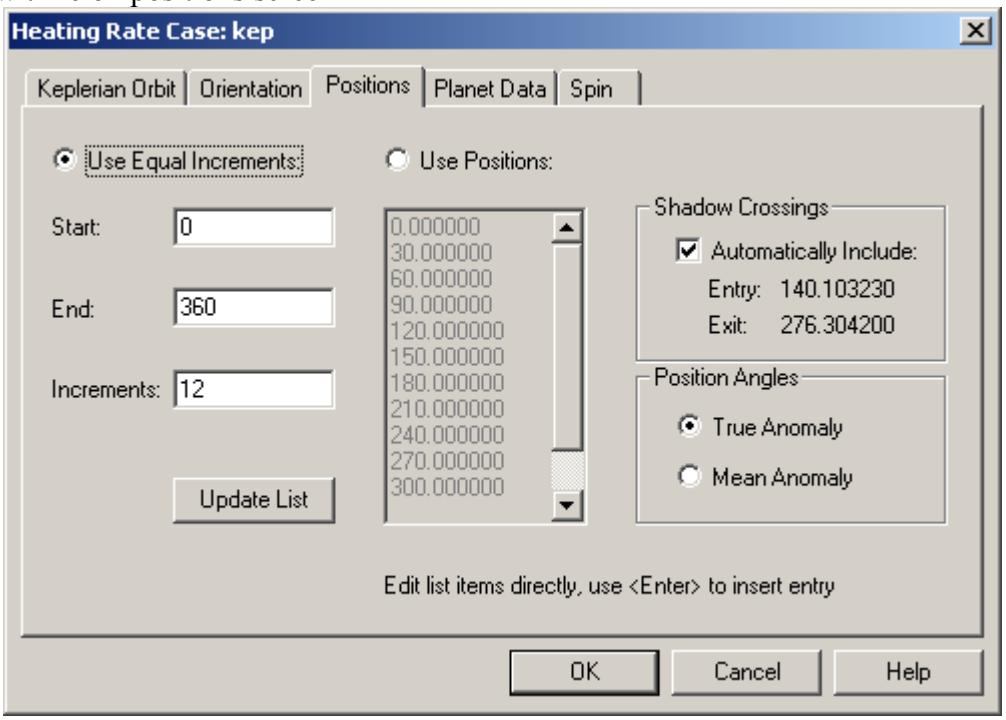
Type in case name based on orbit

Add orbit parameters from STK Excel file (bring across from IDC_Share)

Orbit parameters screen:

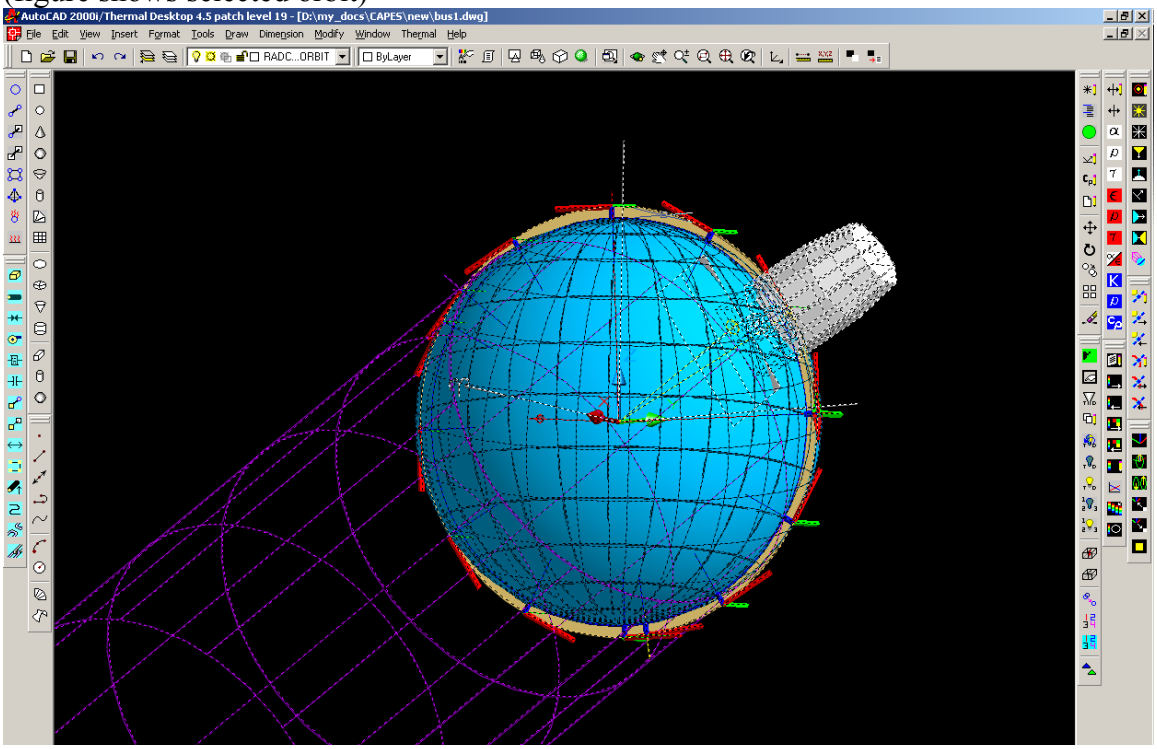



Check shadow time on positions screen



Three ways to confirm orbit with STK person:

1. Compare shadow times
2. Display orbit, then select and type ‘list’ to get vectors to sun and planet at each time point (figure shows selected orbit)



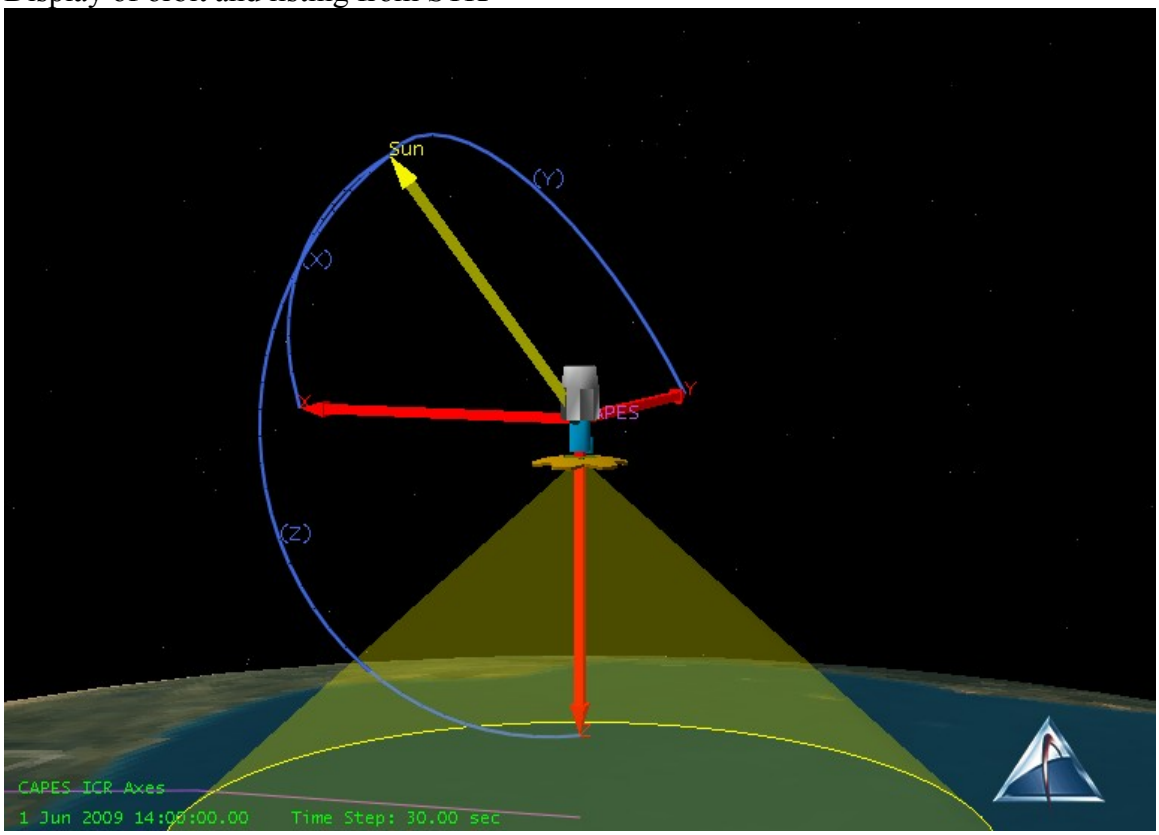

Output of ‘list command:

AutoCAD Text Window - D:\my_docs\CAPES\new\bus1.dwg

Edit

```
1 found

     Data to be used for heating rate calculations:

Time,  mean,  true,       Illum  Sun Vector              Planet Vector
 Orbit Radius
         0,    0.00,     0.00 sun,   < 0.429,-0.420,-0.800>, < 0.000, 0.000,
1.000>, 1.0627

   462.802,   30.00,    30.00 sun,   <-0.028,-0.420,-0.907>, < 0.000, 0.000,
1.000>, 1.0627

   925.604,   60.00,    60.00 sun,   <-0.478,-0.420,-0.771>, < 0.000, 0.000,
1.000>, 1.0627

   1388.41,   90.00,    90.00 sun,   <-0.800,-0.420,-0.429>, < 0.000, 0.000,
1.000>, 1.0627

   1851.21,  120.00,   120.00 sun,   <-0.907,-0.420, 0.028>, < 0.000, 0.000,
1.000>, 1.0627

   2161.34,  140.10,   140.10 entry, <-0.842,-0.420, 0.338>, <-0.000, 0.000,
1.000>, 1.0627

   2314.01,  150.00,   150.00 shade, <-0.771,-0.420, 0.478>, <-0.000, 0.000,
1.000>, 1.0627

   2776.81,  180.00,   180.00 shade, <-0.429,-0.420, 0.800>, <-0.000,-0.000,
```

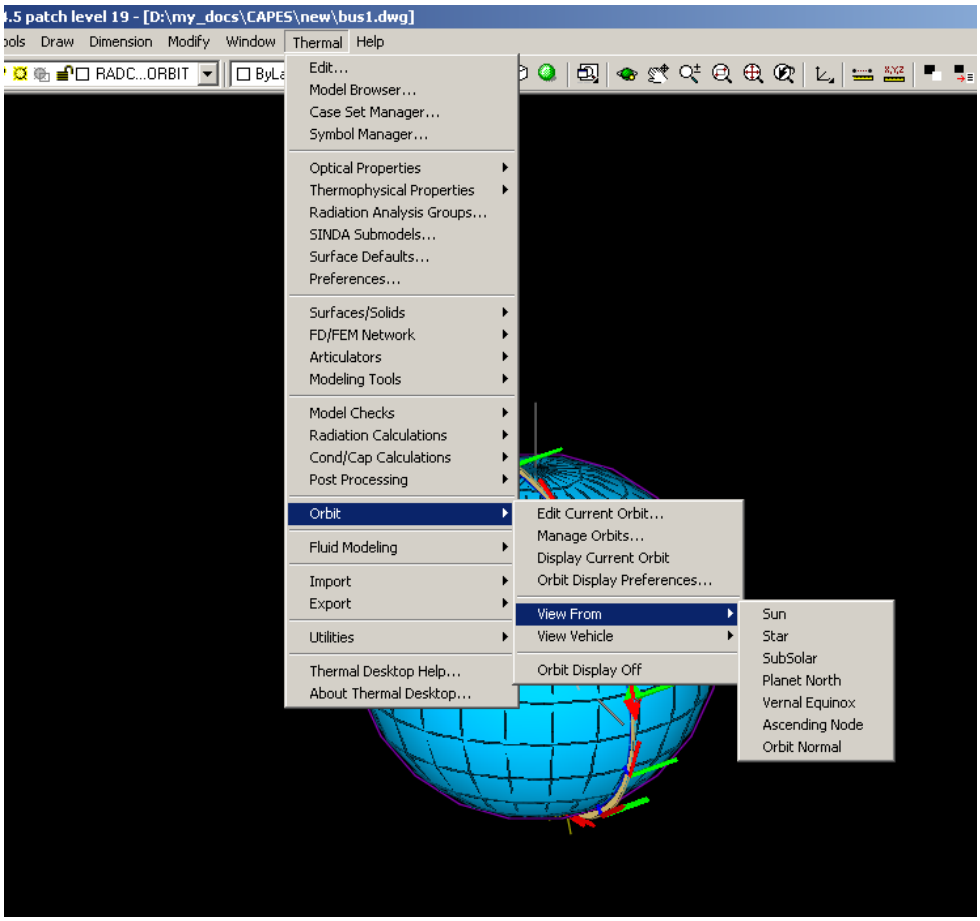
Display of orbit and listing from STK



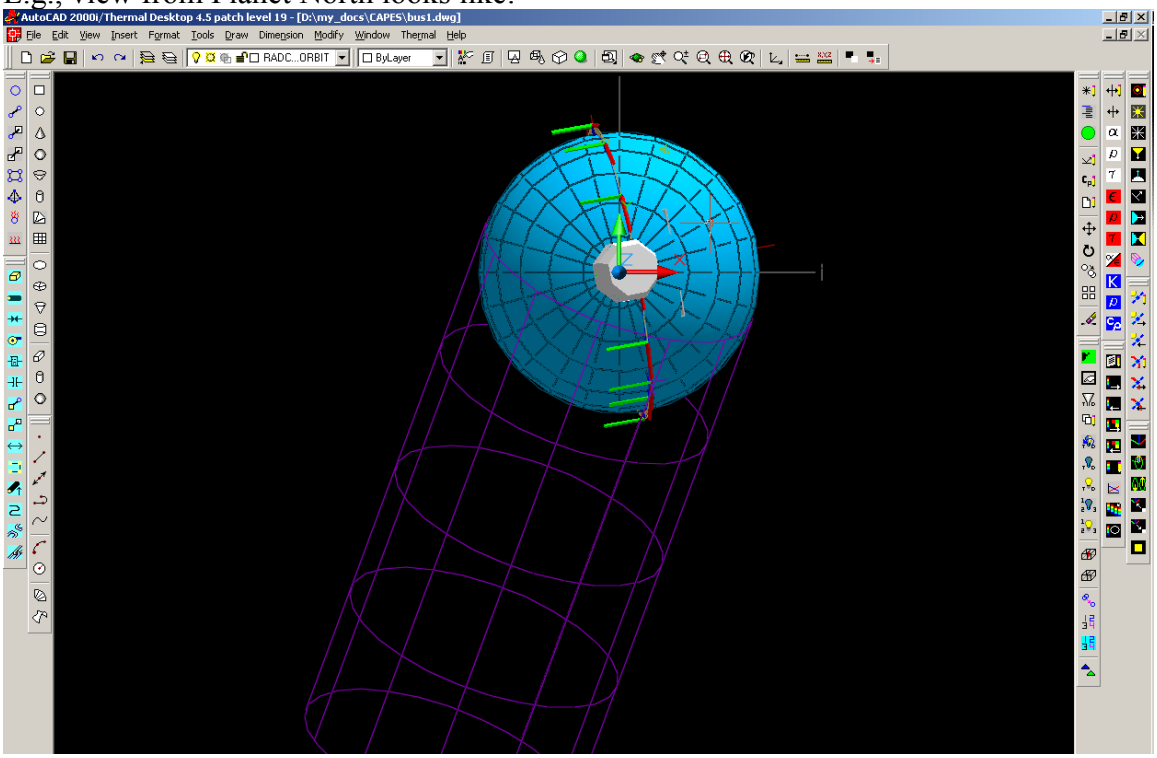

```
Satellite-CAPES:  Lighting


Sunlight Times
--------------
                     Start Time (UTCG)          Stop Time (UTCG)
Duration (sec)
                     ----------------------     ----------------------
--------------
                     1 Jun 2009 14:00:00.00     1 Jun 2009 14:36:08.70
2168.702
                     1 Jun 2009 15:11:07.91     1 Jun 2009 16:08:49.95
3462.038
                     1 Jun 2009 16:43:49.18     1 Jun 2009 17:41:31.19
3462.007
                     1 Jun 2009 18:16:30.46     1 Jun 2009 19:14:12.43
3461.977
                     1 Jun 2009 19:49:11.73     1 Jun 2009 20:46:53.68
3461.947
                     1 Jun 2009 21:21:53.00     1 Jun 2009 22:19:34.92
3461.917
                     1 Jun 2009 22:54:34.28     1 Jun 2009 23:52:16.16
3461.887
                     2 Jun 2009 00:27:15.55     2 Jun 2009 01:24:57.40
3461.857
```

3. Display orbit as “Display from…”
    - Sun
    - Star
    - Subsolar
    - Planet North
    - Vernal Equinox
    - Ascending Node
    - Orbit Normal

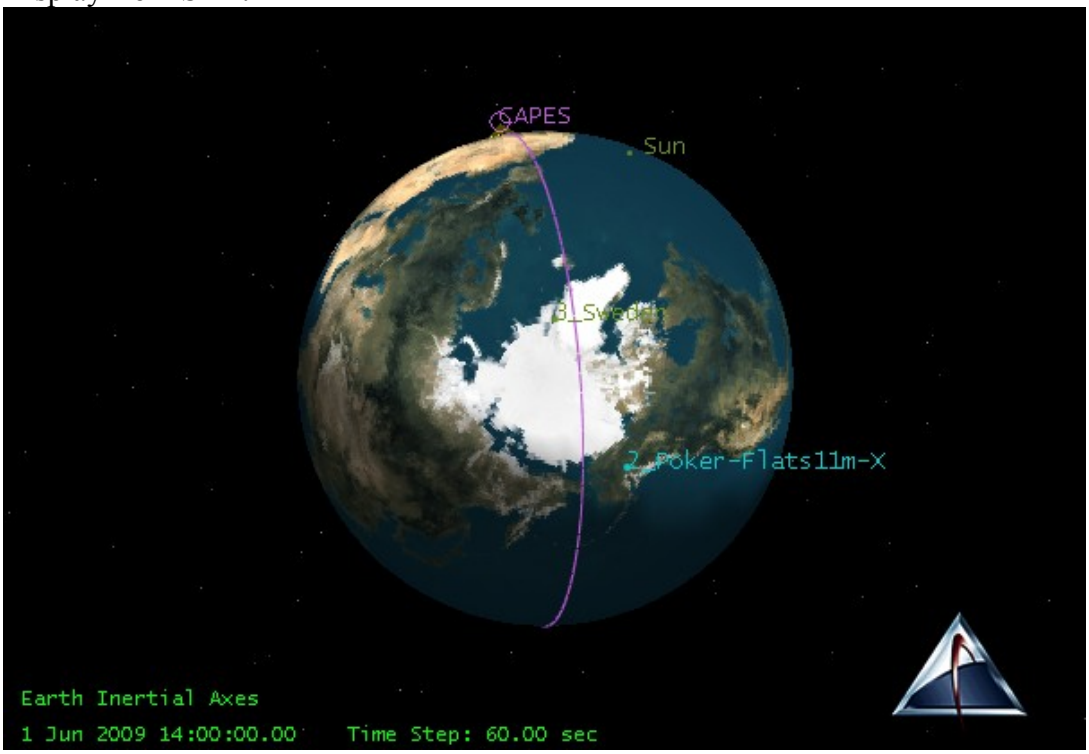
    Use the same display viewpoint within STK and compare visually.



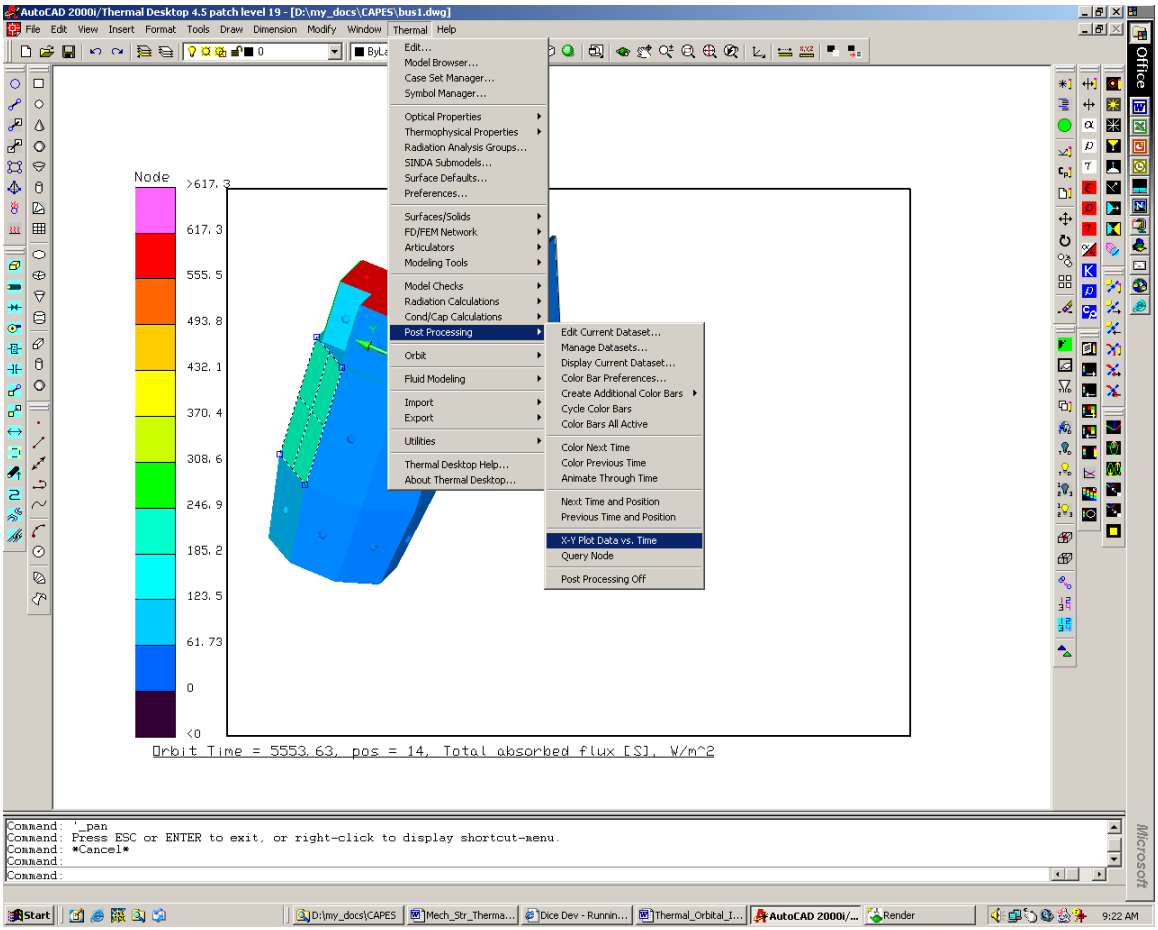
E.g., view from Planet North looks like:



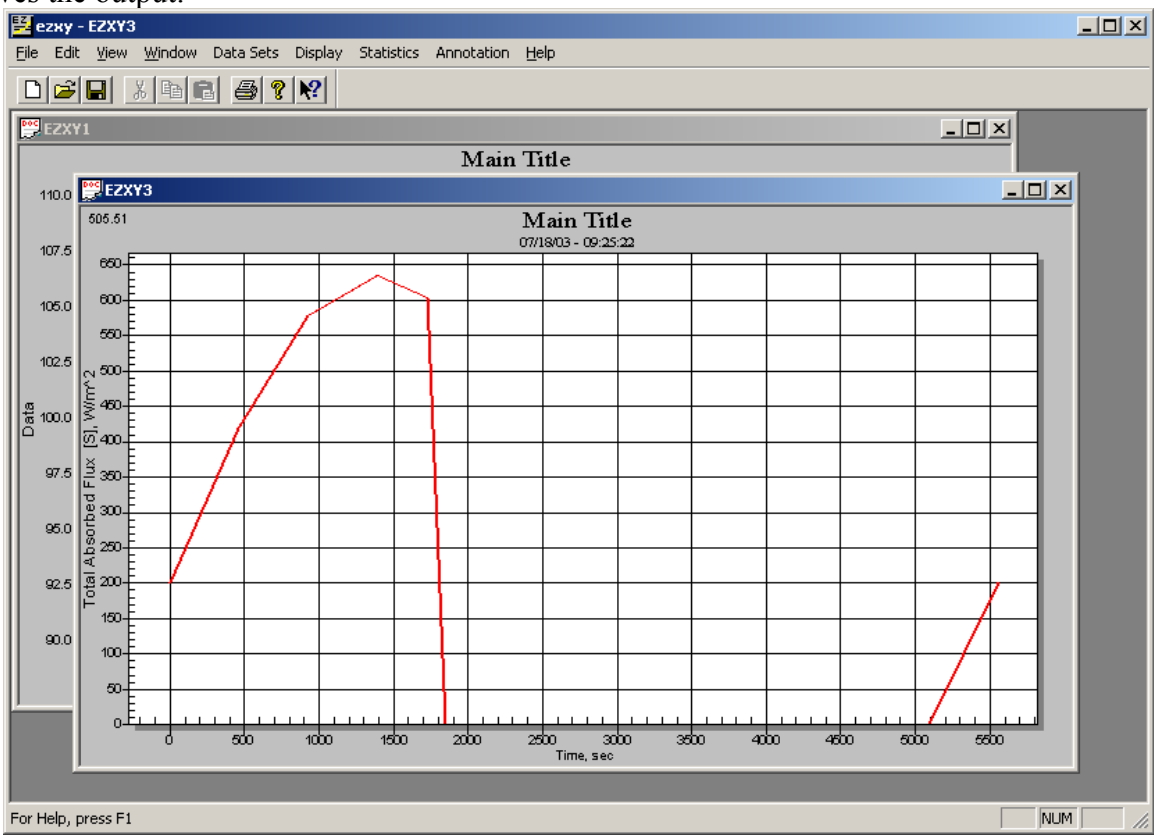

Display from STK:



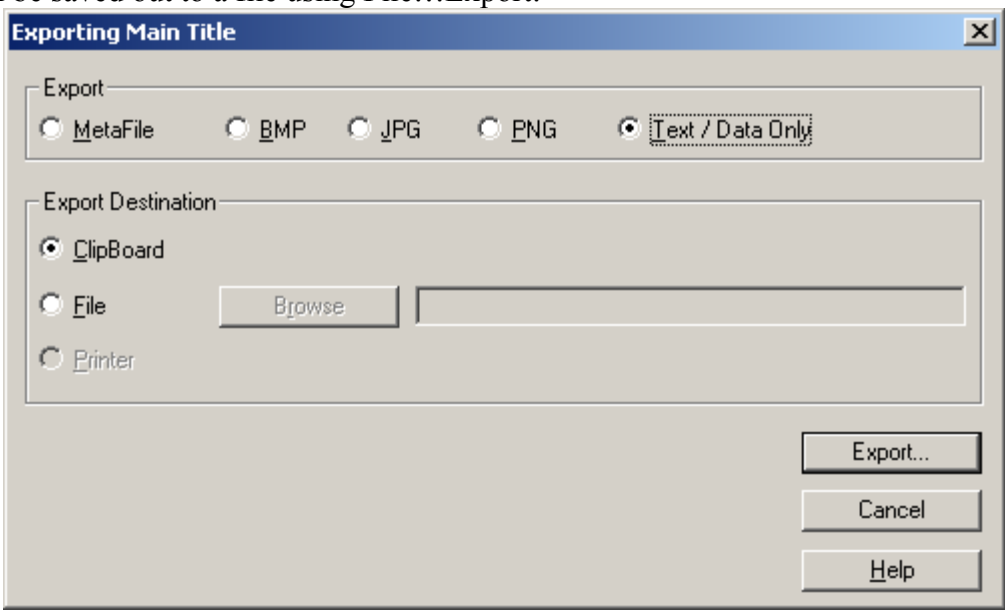
In addition, the flux on a particular node over time can be plotted or saved to a file.
To do this, select the node to be plotted while in post-processing mode, while the data you want plotted over time is post-processed on the model.
E.g., shown is a node selected, and then you select Thermal…Post Processing…X-Y Plot versus Time.



This gives the output:




This can then be saved out to a file using File…Export:




This allows saving to a text file or to the clipboard so that the data can be manipulated in an external program such as Excel.

# Aerothermal Import to Thermal Station

Three examples of transfer from aerothermal to thermal will be described. These are: transient heating uniformly on a face; transient heating that varies across a face, applied as a heat flux; and transient heating that varies across a face, applied using the $h_c$ heating coefficient. All the thermal modeling has been done in the software MSC.Patran, with the thermal analysis solver MSC.Patran Thermal. Patran Thermal currently allows the heat flux result to be pulled in from an analysis run, so that the heat flux applied can be plotted directly on the model.

## *Spatially uniform transient aeroheating*

Heat was applied to the front face for each of several trajectories. The heating for each trajectory was available for the POST runs made by Paul Tartabini. These heating numbers were decreased by a factor of 0.38 (estimated by Joe Olejniczak of JPL/Ames) to account for the fact that Paul T's values were for a sphere of radius 1 m, and the true heating should be for a flat face. This was done by entering the transient in the Heat Flux boundary condition, and adding the 0.38 as a factor. This boundary condition was applied to all the elements on a single face. Example results are shown in Figure 1.

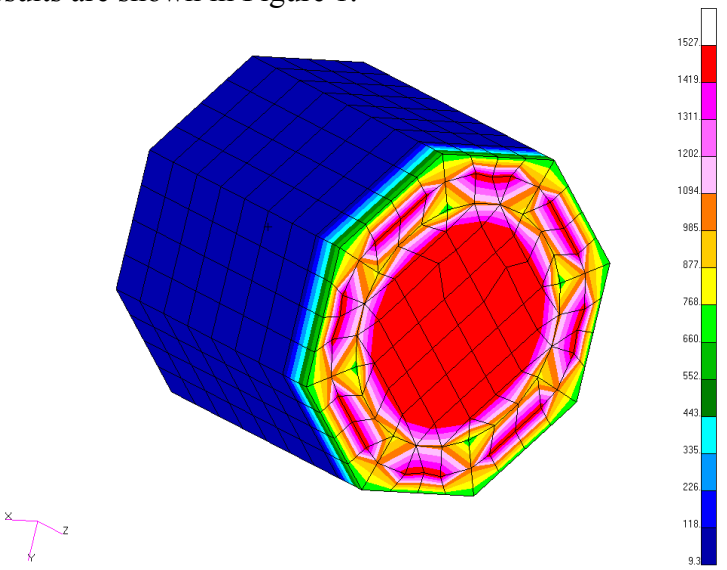



**Figure 1. JIMO Configuration C0 thermal results (traj=450s), at 340s, °C.**

## *Spatially varying transient aeroheating applied as a heat flux*

A model in MSC.Patran was developed, based on the geometry provided in Pro/Engineer. This model was in cm. The Pro/E geometry was imported, which provides B-rep solids ("white solids") in Patran. These needed to be changed to native solids ("blue solids") in Patran, in order to allow hex meshing with the necessary degree of detail. White solids can only be meshed with a tetrahedral mesh. It has been found in past models that a high surface heat load on a tet mesh will not lead to an acceptable solution. Thus, the solids were re-created as native blue solids. The materials were created in the model and applied as shown in Figure 2.

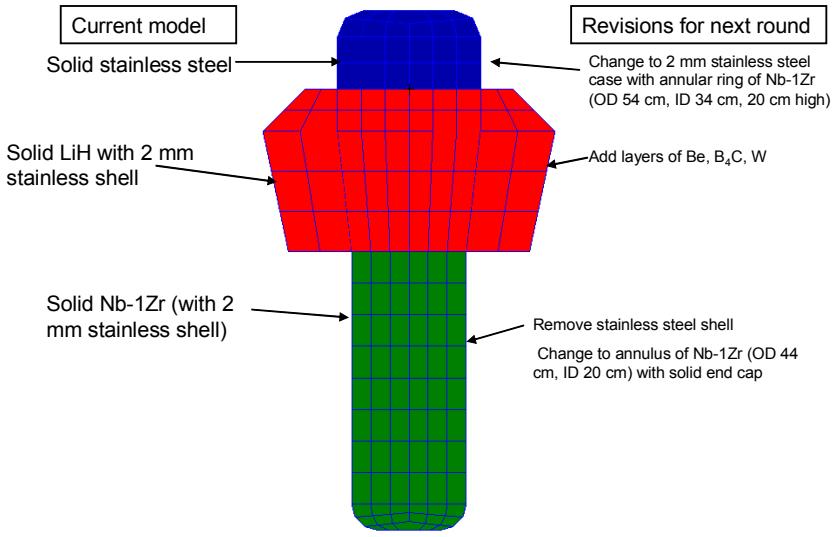



**Figure 2. Initial Configuration D thermal model.**

The file provided by Joe O. for a spatial gradient of heating over the reactor was in terms of r and z coordinates, in cm. In order to use this spatial gradient, a new coordinate system was created in the model, using r-theta-z. (In future analysis, the origin of this coordinate system should be aligned with that of the aeroheating model). In order to have this gradient be a field that would not change between trajectories, it was divided only by Joe's sqrt(rho)$V^3$ value. Then, the spatial field in Excel was broken into sections that were dependent on r, and sections dependent on z. This allows the fields to be applied as 1D fields, which are much simpler to format and apply. Thus, the boundary condition consisted of an r1 field (the smallest radius disk), an r2 field, and a z field (for most of the reactor). In general, as much as possible of the data was put into the z field, and an r field was only used when z was constant. This r (or z) dependent field then had r in cm, and a unitless q factor that was not dependent on the trajectory. Each of these fields were saved as comma separated variable (csv) files, and brought into Patran using a custom utility for field import (bwm_spatial_field).

This method was appropriate for the D0 and D180 cases. For the D90 case, however, Joe O's data was x-y-z dependent (3D), rather than being axisymmetric. In this case, the model was reduced to a quarter geometry model. The 3D spatial field was brought in using a custom utility written by Mike Lindell. First, the node positions on the surface of the model were output to a text file, formatted as *Node ID, x, y, z*. Then the spatial information from Joe was translated to the same coordinate system, and a text file was produced with the format *ID number (arbitrary), x, y, z, q*. Then using Mike's utility (interp3D), the heating data was interpolated on to the Patran mesh. This result was imported and displayed as a color map, and a FEM field created from the values. This field was then applied as the spatial field in the heating boundary condition. In all of this, dimensions were in cm.

This spatial gradient was then multiplied by a time function, which gave the behavior of the heating over time. In order to incorporate Paul T's transient, the value of heating at each time was divided by the peak heating value, giving a normalized, unitless, transient of q/qpeak versus time. This was then multiplied by the sqrt(rho)$V^3$ from the peak heating point in Paul's trajectory (this microfunction is then in $W/m^2$). This was used by taking this modification of Paul T's time-based transient of heating from Excel and placing it as an array in a Patran user file

called micro.dat.apnd. This allows the time-based function to be called in an analysis, and to be easily changed for different runs for the different trajectory cases (by manipulation of a text file). The heating boundary condition applied used the spatial field multiplied by the time microfunction. This was applied as Element Uniform (which applies a constant value across each element), rather than the preferred method of Element Variable (which varies the value across an element), simply because Element Variable does not plot properly (although it does work). Figure 3 shows an example of the spatial factor for the applied heating. In the boundary condition form, the spatial field is given as the Heat Flux Muliplier, and the Template ID calls the appropriate line in template.dat.apnd. The template ID in that file calls the appropriate microfunction in micro.dat.apnd. When the analysis is run, the cm units of the model geometry are translated into m, so that the solution is done in m.

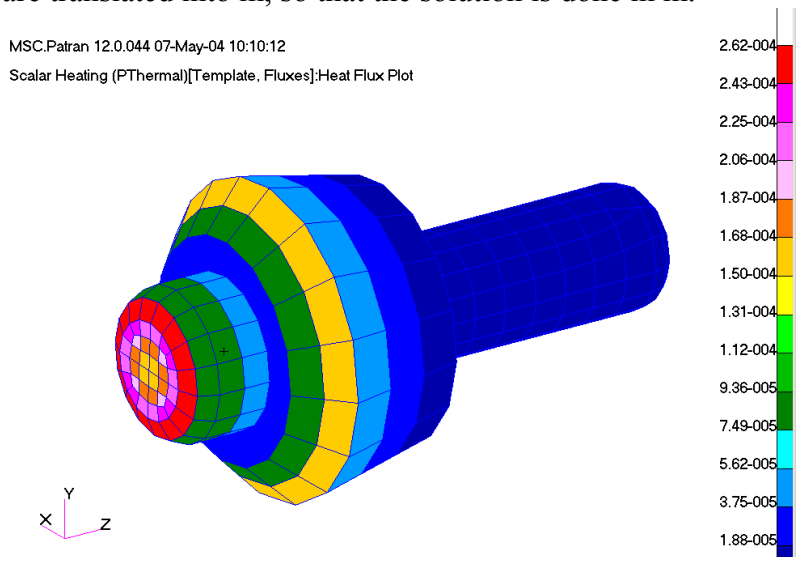



**Figure 3. Applied heating factor boundary condition.**

## *Spatially varying transient aeroheating applied using hc*

Based on this first set of analyses, many refinements were made to improve the analysis for the final set of aerothermal loads. The differences in the analysis consisted of changes to geometry, materials, mesh density, aeroheating application methodology, and transient case definition. The model was redone to more accurately represent the actual structure; the changes described in Figure 2 were made. The resulting model is shown in Figure 4. The mesh density was increased, with a node-to-node distance on the outer surface of about 5 cm, and a thickness into the reactor or about 3 cm. There were about 21000 nodes in the full model. Run time for this model was still only about 5 minutes, so mesh density could be increased further if desired.

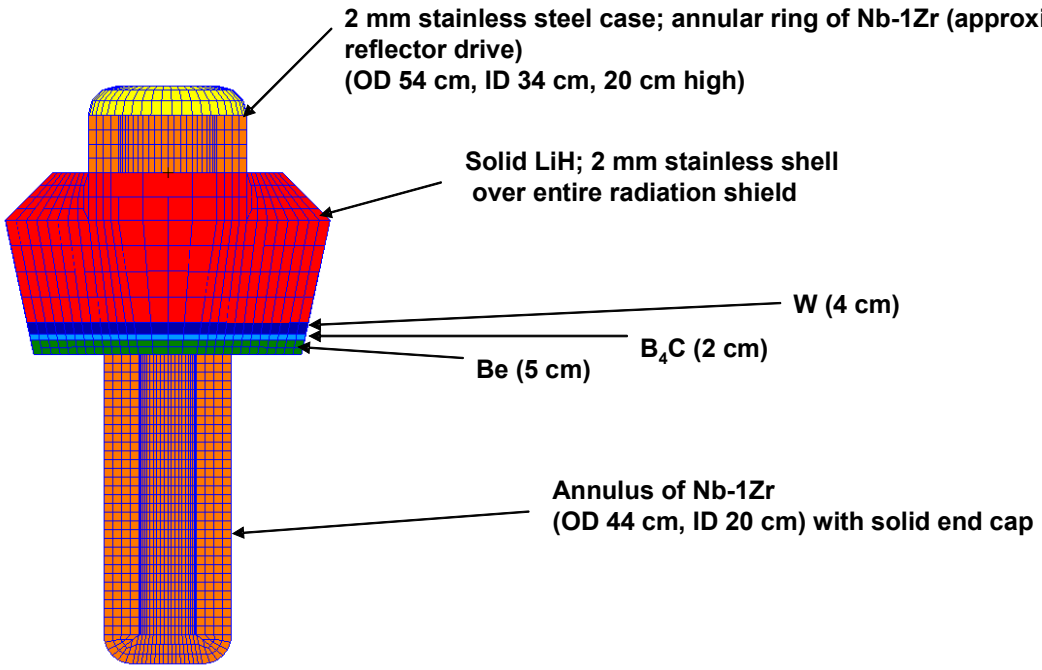



**Figure 4. Revised Configuration D Brayton reactor model.**

The method for application of aeroheating values was changed substantially. In this case, the heating data provided by Joe O. was cold wall, meaning that it was calculated assuming that the entire model surface was at 300K. This will not give correct heating values since the surface will quickly be at a higher temperature than that. It also will not give the correct spatial factor for heat flux, even qualitatively, since the surface will have a temperature gradient, and will not be at a uniform temperature. The surface temperature affects the heat flux, since the heating is proportional to the difference between fluid temperature and skin temperature according to the following:

$$q = h_c * \left(T_{fluid} - T_{skin}\right) \quad (1)$$

where q is heat flux per unit area, and $h_c$ is the convective heating coefficient.
Since Joe's heating was now cold wall, it was not usable directly, as the calculated heating values would be too high, and the gradient would be false. Thus, the decision was made to change the application of heating such that heat would be applied using $h_c$, and the program would calculate heating using equation (1). Thus, on the new model, all boundary conditions were changed to convective. A method for multiplying the spatial gradient by the time-dependent transient was still necessary. The spatial field from Joe thus needed to be transformed to be a heating coefficient field, as described in the following excerpt from Joe:

> Physically, the reference heat transfer values that I gave you need to be scaled to account for the variation in freestream conditions and to account for wall temperature effects. I think in terms of enthalpy, so I'll start with using h:
>
> q(s,t) = qref(s) * [(sqrt(rho_inf(t)))*(V_inf(t))^3] / [(sqrt(rho_ref))*(V_ref)^3] * [h_inf(t) - h_wall(s,t)] / [h_inf_ref - h_wall_ref] * h_inf_ref / h_inf(t)
>
> where
>
> q(s,t) is the time varying heat flux along the surface
> qref(s) are the distributed reference heating rates in my spreadsheets

rho_inf(t), V_inf(t), and h_inf(t) are the time varying density, velocity, and enthalpy along the trajectory
rho_ref is the freestream density used in my qref calculations = 0.00439 kg/m3
V_ref is freestream velocity used in my qref calculations = 5852 m/s
h_wall(s,t) is the time varying wall enthalpy along the surface
h_inf_ref is the freestream total enthalpy used in my qref calculations
h_wall_ref is the wall enthalpy used in my qref calculations, based on Twall=300K.

Now since we don't seem to have the freestream density it Paul's trajectory files, we can use the following

$[(\sqrt{\rho_{inf}(t)})*(V_{inf}(t))^3] = q(t)/q_{peak} * \sqrt{\rho_{peak}}*(V_{peak})^3$

where

q(t) is the heating rate given in Paul's trajectory file
q_peak is the heating rate at the peak heating along the trajectory
rho_peak is the freestream density at the peak heating point along the trajectory
V_peak is the freestream velocity at the peak heating point along the trajectory

The needed enthalpy values can be computed (approximately) in the following ways

$h_{inf}(t) = 0.5*(V_{inf}(t))^2 + Cp_{inf}*T_{inf}(t)$
$h_{wall}(s,t) = Cp_{wall}*T_{wall}(s,t)$
$h_{inf\_ref} = 0.5*(V_{ref})^2 + Cp_{inf}*Tinf_{ref}$
$h_{wall\_ref} = Cp_{wall}*T_{wall}$

where

Cp_inf is the specific heat at constant pressure of the gas in the freestream
Cp_wall is the specific heat at constant pressure of the gas at the wall
T_inf(t) is the freestream temperature
T_wall(s,t) is the wall temperature along the trajectory

We now also make the following approximations:

Cp is constant, so all Cp values are the same = 1004.84 J/kg/K
Assume T_inf is fixed at 300 K
Define $Cp * T_o = 0.5*V^2$

Then the term

$[h_{inf}(t) - h_{wall}(s,t)] / [h_{inf\_ref} - h_{wall\_ref}] * h_{inf\_ref} / h_{inf}(t)$

becomes

$[T_{o\_inf} + Tinf - T_{wall}(s,t)] / [T_{o\_inf} + T_{inf}] * [T_{o\_ref} + T_{inf}] / T_{o\_ref}$

where $T_{o\_inf} = 0.5*(V_{inf}(t))^2 / Cp$

Finally, we can write

$q(s,t) / \{Tfluid - T_{wall}(s,t)\} = qref(s) * [\sqrt{\rho_{peak}/\rho_{ref}}] * (V_{peak}/V_{ref})^3 * [q(t) / q_{peak}] * [0.5 * (V_{ref})^2 / Cp + T_{inf}] / [0.5 * (V_{inf}(t))^2 / Cp + T_{inf}] / [0.5 * (V_{ref})^2 / Cp]$

where $Tfluid = [0.5 * (V_{inf}(t))^2 / Cp] + T_{inf}$

Basically, the spatial field supplied by Joe was used, and all the modifications to that value that made it a correct heating value based on the skin temperature were lumped (by Joe) into a transient microfunction Hbar(t). This changes the q value to an $h_c$ value. Lumped into the Hbar(t) factor are the ratios to factor Paul and Joe's different trajectory sqrt(rho)$V^3$ values, the factor to account for the cold wall temperature, and the q/qpeak ratio to give the transient behavior of Paul's trajectory.
The spatial q field from Joe was brought in as described in the previous section, using r and z dependent fields for D0 and D180, and 3D fields where the heating was not axisymmetric (D90, D10 and the tumbling case). The temperature of the fluid was brought in as a transient microfunction, different for each of the three new trajectory cases from Paul (ballistic coefficients of 4406, 3470, and 9252 kg/$m^2$). The transient factor Hbar(t) provided by Joe as the appropriate multiplier for the spatial values was constructed in Excel, multiplied by the 1.5 factor recommended by Joe O. to account for turbulence. This array was placed in the Patran user file mat.dat.apnd, as that is where transient convective factors are maintained.
In this case, the boundary condition applied was Convection, Template. The spatial field was placed in the box "CONV GP2/GP3". The Template ID was given in the appropriate box, as was the fluid temperature node ID. The template ID line in template.dat.apnd pointed to the microfunction (in mat.dat.apnd), and used convection configuration 39, which is used in Patran for multiplication of a spatial field and time field.
One item of interest is that when this change (to aeroheating application via $h_c$) was first made, the thermal results were completely different than the original. This was traced back to a bug in the Patran Thermal software, which did not perform a correct multiplication of time-dependent and spatially dependent fields in the convective heating boundary condition. This bug was fixed in real time by MSC, and solution could proceed.

Examples of the solutions are given in the following figures. As before, each time a different configuration was run, the results were brought into Patran, and the maximum temperature node for each material block was determined. Then the transient behavior of those nodes was plotted from the results. One other small item of interest on the Patran Thermal software is that during the time span of this analysis, a new feature of heat flux plotting was implemented in Patran Thermal. This allows plots of the heat flux over the mesh, as shown in Figure 6. However, there appears to be an additional bug in the Patran Thermal code, which is not allowing correct plotting of heat fluxes for fields that were applied with symmetric r and z dependence. This problem is currently being worked.

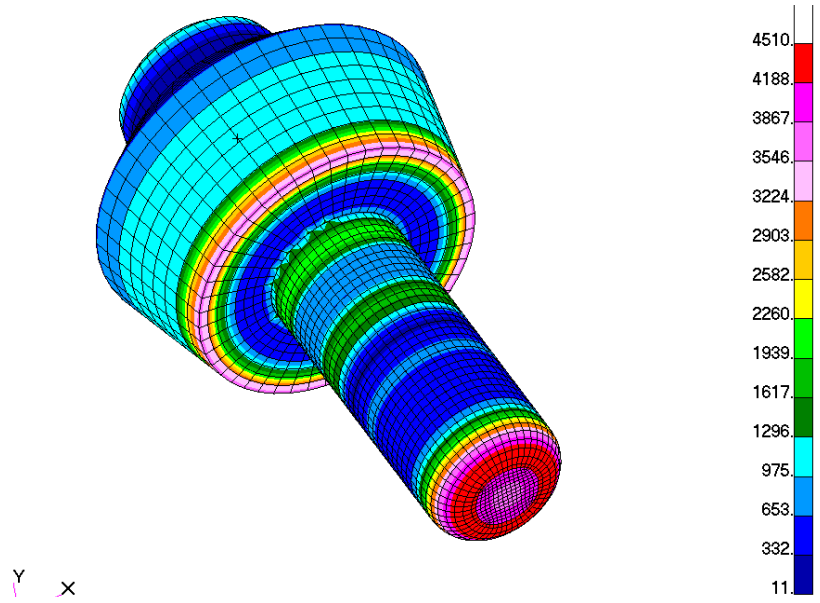



**Figure 5. Configuration D0, bc 9252, time 505s, °C.**

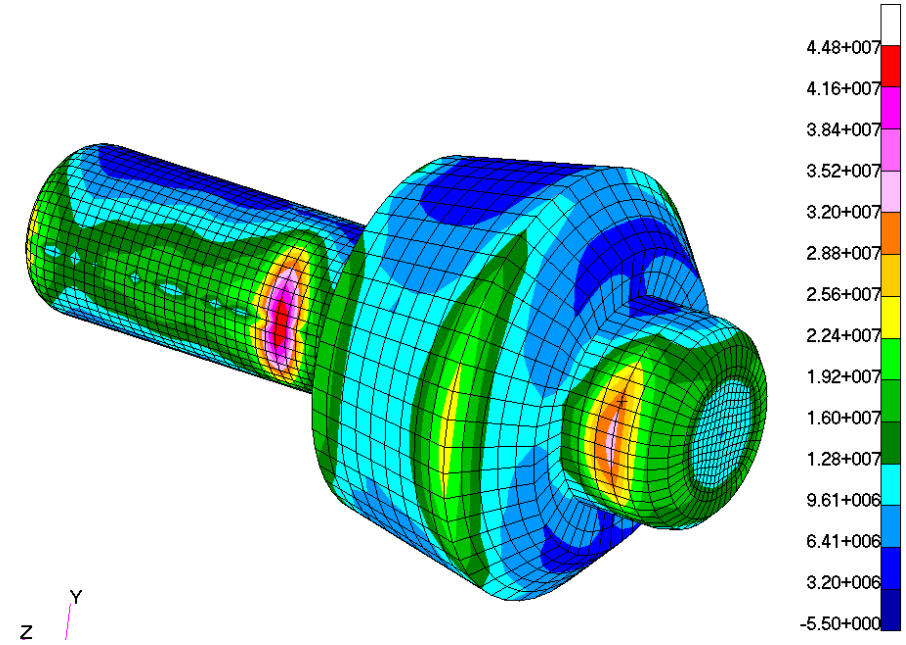



**Figure 6. Heat flux for Configuration D tumbling, bc 4406 case (3D) with 1.5 factor, at 505 s, W/m$^2$.**

## *Aeroheating applied using ulib fortran (spatial variation changes over time)*

### ***Hyper-X example***

In this example, the aeroheating was not a constant gradient that could be multiplied by a transient factor. The gradient was complex, and changed at each time point. Thus, specialized code was written within the user-customizable portion of Patran Thermal (ulib) to allow

implementation of this type of heating. This software is available for use at other NASA Centers if desired.

The aeroheating was applied using two different methods for different regions. Detailed description of these methods can also be found in earlier papers[i,ii,iii,iv,v,vi]. For the area of the actual sharp leading edges, the stagnation point heating was calculated via Fay-Riddell methods using an internal Langley program called stagheat.exe. On the flat body sections (acreage), the aerodynamic heating loads from the hypersonic airflow were predicted by the Supersonic Hypersonic Arbitrary Body Program (SHABP)[vii]. Both the stagnation point and SHABP heating loads were only computed at about 20 discrete time steps along the 130-second trajectory, which was a much larger time step than would be used in the thermal analysis. The trajectory began at the time the Hyper-X vehicle was dropped from the B-52. The mesh used in the SHABP model was much coarser than the thermal model would need. Thus, the thermal analysis needed to provide accurate interpolation of the heating data (both in space and time) onto the thermal model. Both stagheat.exe and SHABP provided maps of heat flux (q) over the specified area at each given time point. These heat fluxes were obviously dependent on the skin temperature of the tail. Thus, after a solution was run for a given set of q maps, the temperature maps were provided back to SHABP and stagheat.exe, and new q maps based on the latest temperature prediction were run. Several (3-4) iteration loops were usually required for this cycle to come to closure. Closure was defined as when temperatures between solution sets were varying less than 10°F.

An alternative method was used on the acreage loads for vertical tail, in order to minimize this iteration time to come to closure. In this method, the SHABP model was run for several uniform wall temperature sets on the entire component, rather than run for a specific predicted temperature gradient. Then, within Patran Thermal, the solver interpolated on temperature as an additional variable, to find the heat load for any given node based not just on the problem time and its location, but also on its temperature at that time. This allowed the problem to be solved in a single run of each program, rather than iterating between them. It also allows for a more detailed map of the skin temperatures to be used to compute the heat flux, and so in regions of sharp thermal gradient, it gives a better representation of the heat flux.

Incorporation of the stagheat.exe loads on the leading edges was fairly straight-forward. Temperatures and body angles for several representative points on each leading edge were used in stagheat.exe to generate a file of heating versus time for each point. A file in PATRAN format of time-dependent heating based on different body angles along the leading edge was produced. To apply that heating in Patran, a product of functions was used, as shown in equation (1). These determined the difference in temperature between the node the heat is applied to, and the representative node used to calculate the heating at that body angle, to correct the heating for any given node based on its temperature. What was used in PATRAN is the exact heating for each node based on its temperature, $Q_{node}$, where

$$Q_{node} = Q_{ref} * \left( \frac{\left(T_{ext} - T_{node}\right)}{\left(T_{ext} - T_{ref}\right)} \right) \tag{1}$$

and where

$$Q_{ref} = h * \left(T_{ext} - T_{ref}\right)$$

Incorporation of the SHABP aeroheating loads was more complex. The SHABP loads were calculated based on the aeroheating grid, a different mesh than the PATRAN model, at 18 time points. These files were pulled into the PATRAN run and interpolated in space and time using user-developed software within a routine provided in PATRAN called ulib. This ulib could include user modifications to subroutines that were called at specific times during the solution. For each node that had an aeroheating boundary condition applied, the thermal solution invoked a umicro.f routine within ulib. This routine determined the nodal position and calculated the aeroheating value via a weighted interpolation of four points in the SHABP grid. Since the SHABP grid was highly swept, and the PATRAN grid was very fine but not regular, being comprised of tetragonal elements, this calculation was not trivial. Due to the high sweep angle in both the SHABP grid and the leading edge itself, and the sharp decrease in aeroheating with distance away from the leading edge, the four points selected might not be the nearest to the node, but needed to bracket it in terms of distance from the leading edge. The interpolation could not be done in the orthogonal x-y axes, but had to be in a system orthogonal to and parallel to the leading edge. This coordinate system was obviously different for the upper and lower leading edges.

Due to geometry considerations, some modifications needed to be made to the basic aeroheating flux predicted by SHABP. These involved load amplifications due to geometry elements such as corners, gaps and shock wave off other parts that were not considered by SHABP. SHABP is an engineering code, not a full CFD code. The use of full CFD in this case, with multiple time steps to be analyzed for several different configurations, was not feasible. The most challenging shock considerations were for the intersection between the vertical tail and horizontal wing. Depending on the vehicle speed, angle of attack (AOA), and the angle of the wing, there could be a shock wave off the leading edge of the wing onto the vertical tail in either the upper or lower direction, or both. Shock occurred only on the outboard surfaces of the tail (adjacent to the wing), not on the inboard surfaces adjacent to the fuselage.

An example of the thermal results from the vertical tail is shown in Figure 7. The effect of the high stagnation heating at the leading edge can be clearly seen, as well as the effect of the internal cavities with thin skins heating more than the solid areas.

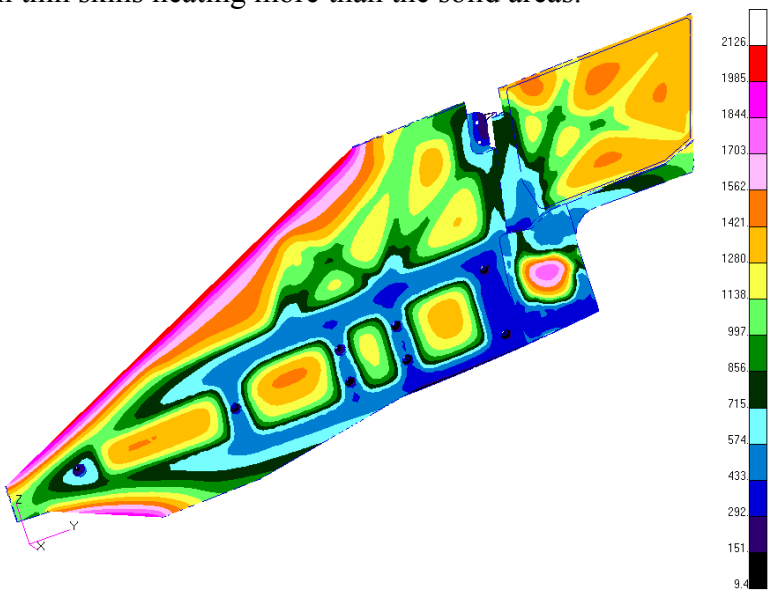



**Figure 7. Example temperature distribution at end of trajectory on vertical tail (°C).**

[i] Amundsen, Ruth M.: “Method Improvements in Thermal Analysis of Mach 10 Leading Edges,” presented at the Tenth Conference, Workshop and Product Presentation on Thermal and Fluids Analysis Tools and Methods, NASA Huntsville, AL September 13-16, 1999.

[ii] Amundsen, Ruth M.: Comparison of Integrated Analysis Methods for Two Model Scenarios, Ninth Conference, Workshop and Product Presentation on Thermal and Fluids Analysis Tools and Methods, Cleveland, Ohio, August 31--September 4, 1998.

[iii] Amundsen, Ruth M.; and Leonard, Charles P.: "Hypersonic Thermal Analysis including Shock Interaction Load Amplification with Motion Effects", 27th Annual Conference on Composites, Materials and Structure, January 27-31, 2003, Cocoa Beach, Florida.

[iv] Lindell Michael C.; and Amundsen, Ruth M.: “Nonlinear Thermal/Structural Analysis of Hypersonic Vehicle Hot Structures”, presented at the NASA Workshop on Innovative Finite Element Solutions to Challenging Problems, NASA GSFC, May 2000.

[v] Amundsen R.M.; and Torres, A. O.: “Thermal Analysis of the Hyper-X Research Vehicle Wing: Mach 7 Design,” presented at the 97 JANNAF Conference, Chemical Propulsion Information Agency, West Palm Beach, Florida, October 27-31 1997.

[vi] Amundsen Ruth M.; and Brzowski, Matt: “Correlation of Thermal Model with Hyper-X Leading Edge Material Testing,” presented at the 25th Annual Conference on Composites, Materials and Structures, Cocoa Beach, Florida, January 22-26, 2001.

[vii] Gentry, A.E.; Smith, G.N.; Wayne, Oliver: The Mark IV Supersonic Arbitrary Body Program, AFFDL-TR-73-159, Volumes I, II & III, November 1973.